ROUGH TURNING THIS 18-TON CAST-IRON DRIER ROLL IS ACCOMPLISHED IN ONE CUT AND WITH ONE INDEX OF THE INSERTED-TIP CARBIDE TOOL

Courtesy of Kennametal Inc

CARBIDE CUTTING TOOLS

How To Make and Use Them

by

Warren Baker, M.S.

Executive Editor, Retired, *Science and Mechanics;* Managing Partner, Model Engineering Company, Chicago; Member, American Society for Metals

and

Joseph S. Kozacka, B.M.E., M.S.

Associate Professor of Mechanical Engineering and Head of the Engineering Shop Laboratories of University of Illinois, Chicago Branch

Engineering Consultant for Vascoloy-Ramet Corporation

Supervisor, Engineering Science and War Training Courses, Illinois Institute of Technology

Formerly Associate Professor of Mechanical Engineering and Director of Shops, Illinois Institute of Technology

ILLUSTRATED

D. B. TARAPOREVALA SONS & CO. PRIVATE LIMITED

210 Dr. Dadabhai Naoroji Road, Bombay 1

1st Printing 1949
2d Printing 1951
3d Printing 1957
4th Printing 1958
5th Printing 1962
6th Printing 1963

First Indian Reprint, 1970

This book has been published with the assistance of the Joint Indian-American Textbook Programme. It includes every word and illustration contained in the latest original higher priced edition of this title and is a complete and authentic copy thereof.

Price in India : Rs. 8.00

Printed and published by R. J. Taraporevala for D. B. Taraporevala Sons & Co. Private Ltd., 210 Dr. Dadabhai Naoroji Road, Bombay 1, at their Electrographic Industries Division, Apte Industrial Estate, Worli, Bombay 18.

PREFACE

Events in recent years have greatly influenced industrial development with the result that more advances in manufacturing methods have been made during this period than at any time in the history of mankind. Similarly, the role of the cutting tool has increased in importance, and considerable research has been devoted to the quest for tool-bit material of a nature adequate for the demands placed upon it. This search has culminated in the discovery and increasing use and application of the carbides.

The impact of the introduction of carbide cutting tools on the metalworking industries is just now being felt. Although such tools have been known and used to some extent for many years, it is only recently that they have been generally accepted as the superior cutting tool experience has proven them to be.

As might be expected, the wholesale use of this material for tipping cutting tools has created many problems in design, manufacture, use, and maintenance of such tools. Recognizing the need for a dependable guide covering these four cardinal points, the authors have endeavored to gather in one place, and in as simple and straightforward a manner as possible, all the known facts regarding the use of the carbides as cutting tools. Some of the things described in this book have been discussed previously in the highly technical journals of the machine and metallurgical trades, in government publications during wartime, and in the learned dissertations of research engineers. But much of this book is the result of original research and has never been told at all. And never before has this accumulated information been put in one place for ready reference and in a form understandable to the reader not so fortunate as to possess an engineering degree.

Carbide Cutting Tools is primarily intended for the vocational student, the man at the machine, and the tool buyer. However, tool engineers and designers also will find it invaluable as a ready source of information. Considerable space has been devoted to the description of the assembly of tool tips and shanks. Proper grinding procedures are given for all types of tools. Methods of effectively converting older machines to the use of carbide tipped tools are discussed. Care and inspection of newer machines equipped with carbides are described. A comparison is drawn between high-speed tools and the carbides. The different grades of carbide are explained, their names given, and their characteristics recorded.

The book is profusely illustrated with drawings and photographs which are carefully tied in with the text. In addition, considerable important information is made conveniently available in the wealth of tabular matter which is found throughout the book. Three chapters

on design are included for the purpose of acquainting the reader with the problems of design rather than to give hard and fast rules for the solution of problems.

The mathematics of Carbide Cutting Tools has been limited purposely to a minimum. In many instances it would have been simpler for the authors to have used higher mathematics in presenting their subject, but it was felt that the field of carbide tools has been made mysterious and difficult too long. Thus, an overt effort has been made to tear away the veil and put the ability to use the carbides intelligently and well into every man's hand. In the years ahead, everyone connected with the machine tool and metal-forming industries will need that knowledge and ability.

The Publishers

ACKNOWLEDGMENTS

To the following individuals and firms, the authors wish to express their gratitude for making this book and its wealth of illustrations possible:

To Wesley G. Paulson, editor, "Machine Tool Blue Book," Chicago, for his unfailing kindness and aid; E. J. Tangerman, editor, "American Machinist," New York; Ernest E. Thum, editor, "Metal Progress," Cincinnati; and to Athel F. Denham, Detroit publicist and consultant, for their patience and assistance.

To Russell P. MacFall, A.M., Chicago geologist and mineralogist, for many hours spent gathering facts.

To Sam Beall and Charles M. Reesey of Cincinnati Milling & Grinding Machines, Inc., Cincinnati, for data and photographs in profusion.

To John F. Wilson of the Ohio Crankshaft Company, Cleveland; E. Hollingsworth, Vascoloy-Ramet Corp., North Chicago, Ill.; Alex F. Kelberer, The Carborundum Company, Niagara Falls, N.Y.; Miss Bess Herbert, Westinghouse Electric & Mfg. Co., Pittsburgh; J. W. Underhill, Eutectic Welding Alloys Company, New York; R. R. Weddell, Weddell Tools, Inc., Rochester, N.Y.; W. L. Rybolt, the R. K. LeBlond Machine Tool Co., Cincinnati; J. Potter Cunningham, Potter & Johnston Machine Company, Pawtucket, R.I.; J. M. Kjellstrom, Sundstrand Machine Tool Co., Rockford, Ill.; Edward J. Terry, Eclipse Counterbore Company, Detroit; I. F. Yeoman, The International Detrola Corp. (Foster Division), Elkhart, Ind.; John P. Bunce, Kearney & Trecker Corp., Milwaukee; J. P. Burdick, Luther Goff, and R. F. Perkins, Brown & Sharpe Mfg. Co., Providence, R.I.; G. B. Moynahan (Chicago), Haynes Stellite Company, Kokomo, Ind.; Arthur E. Schulz, Ex-Cell-O Corp., Detroit; J. C. L. Brown, Landis Tool Company, Waynesboro, Pa.; Fred W. Pennington, Kennametal, Inc., Latrobe, Pa.; Paul A. Ross, Consolidated Machine Tool Corp., Rochester, N.Y.; R. W. Thomas, McCrosky Tool Corp., Meadville, Pa.; E. C. Howell and John Mason, The Carboloy Company, Inc., Detroit; C. L. Shaw, the Norton Company, Worcester, Mass., and J. E. Craig, the Warner & Swasey Co., Cleveland, for data, photographs, time, and assistance.

To Arthur E. Burke, Head of the Illustration Department of the American Technical Society, and his staff, for carefully prepared and skillfully executed sketches and drawings.

To B. E. Ferrell, Jr., of the American Technical Society, Editorial Staff, for his work in preparing the manuscript for publication.

And to A. C. Barnett, Weston-Barnett, Inc., Chicago, gentleman, counsellor, and friend who started us on the search that brought this book into being, our special thanks.

The Authors

CONTENTS

The Wide Application of Carbide Tools in Industry Has Resulted in This Varied Selection of Tools and Tips

Courtesy of Vascoloy Ramet Corporation

CHAPTER I

Machine Tools and Carbides

Early Development of Machine Tools. The story of metal tools is the story of civilization. Biblical accounts of the genesis of our world and its people are followed in the next chapter by mention of Tubal-cain, "instructor of every artificer in brass and iron." When primitive man laid aside his stone weapons for tools of copper and bronze, the shift in his development was so great that the era became known to science as the Bronze Age. The advent of iron brought similar advance.

If, on the heels of the Iron Age, today and tomorrow go echoing down the corridors of time as the Machine Age, history must give a good share of the credit to an amazing new science, powder metallurgy, and to one of its first products, the sintered carbides. Without question, these new tools of the twentieth century have revolutionized the art of metal working. Although this new science and art performed wonders in the recent war, it is still only on the threshold of its usefulness.

Let us look back a bit to substantiate that statement. The machine tool, in its essentials, is a power-driven device which removes metal "chips" from work material. Whether it be a lathe, planer, drill press, broach, miller, or even a power saw, it can be considered a machine tool, and the machine tool as such is a fairly recent product. It was made possible in the nineteenth century with the development of machines to produce power. James Watt and his steam engine (shown in Fig. 1) may well be given credit for ushering in this period of power and power-driven tools.

At first thought, a knowledge of the history of machine tools and the development of their uses would not seem particularly valuable. Nevertheless, in view of the tremendous changes taking place today, it seems that the designer, the buyer, the user, and perhaps even the man who maintains such tools, might well be expected to know something of their background. For this reason the story of their development will be told in this opening chapter in a more or less summary form.

It is true, of course, that hand- and horse-powered machines were used to form materials long before Watt's time. A form of hand-powered lathe is shown in Fig. 2. Many woodworking machines were operated by hand long after his day. Watt's big difficulty in building his steam engine was in forming the cylinder and keeping the piston tight within it. He

used cork, oiled rags, old hats, burlap, paper, or anything that was handy to wrap about the piston to achieve the result gained by the piston ring of today. Still, there was a serious loss in compression because the diameter of his cylinder varied from end to end by as much as 3/8", which is not surprising in that it was neither bored nor cast, but hammered into shape. In Watt's day (1769) there existed neither the tools nor the men to build properly as simple a machine as a one cylinder steam engine.

Up to the time of Watt, practically all machinery was made of wood with such hand tools as were available. Perhaps some of the fastenings

Fig. 1. An Old Woodcut Depicting James Watt Studying the Model of His "Improved" Steam Engine in His Laboratory

Courtesy of The Bettmann Archive

and smaller parts were made of metal and may even have consisted of a few castings and forgings fitted by hand; but the tool equipment owned by the millwright, as the metal worker was known in those days, consisted essentially of a hammer, chisel, saw, and a file. His only measuring instrument was a wooden rule, and it was prone to vary from shop to shop. Mechanical equipment was much the same as it had been for generations.

Even after Watt, it was not until 1800 to 1840 that the first great period of development took place. In 1820, Joseph Clement built his first planer, a remarkable machine for its day. It was operated by only one man, according to Smiles' Industrial Biography, "though two were

employed to make long and full cuts both ways." Clement enjoyed an almost total monopoly on the planing business for more than ten years. According to Smiles, the charge for planing was 18 shillings a square foot, or about $3.60 at the current rate of exchange. This gave Clement an income of about ten pounds, or $40.00, for each 12 hour day, during which time he was able to cut 11 square feet. Today, with carbide tools, it would be possible to cut 11 square feet of the toughest steel in about five minutes, and the cost would be less than 40 cents. Thus has metal working advanced.

Fig. 2. The Bow Lathe, Used for Making Wooden and Metal Screws. The Workpiece Was Turned in Alternate Directions by the Treadle Below, and by the Bow and String Above
Courtesy of The Bettmann Archive

It is interesting to note that when Clement's planer was the fastest, best, and most efficient machine known, high-carbon tool steel was the only metal cutting material available; yet high-carbon steel is still one of the hardest materials we have for machining purposes. Hardness, however, is not today's only requisite. It was possible in Clement's day to cut at about 25 f.p.m. (surface feet per minute) with such tools. However, this speed caused such heat that the tool soon softened and lost its ability to continue cutting. Even as low a speed as 25 f.p.m. was impractical.

Heat treatment in those days also was largely a matter of guess, and was often a process kept secret by its users. The only means of deter-

mining temperature was by the color of the heated metal. In his book, Super Finish, A. M. Swigart tells how the manager of a small plant once had all the windows washed with the result that the dark corner where the tool hardening department was located was so brightened that the heat treaters could not judge the temperature of the metal. Washing the windows almost caused a shutdown of the factory.

So, even at as slow a speed as 25 f.p.m., tool life 100 years ago was short and the tool itself unreliable. In practically all instances the business of making tools, heat-treating them, and cutting and re-

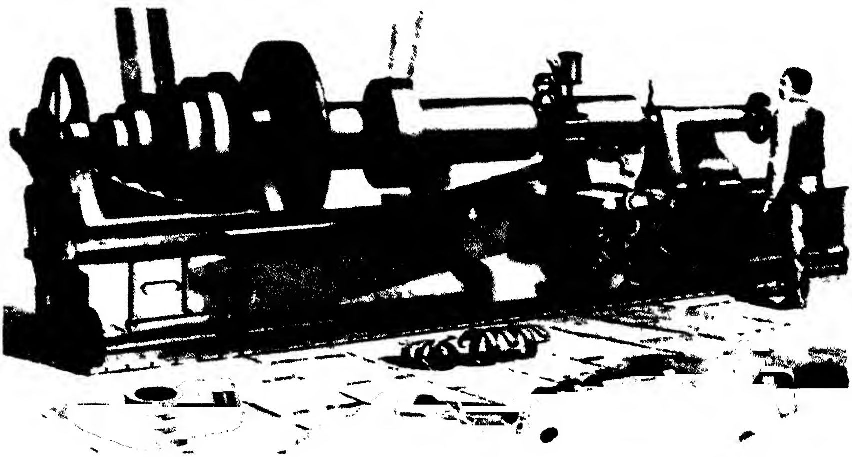


Fig. 3. Lathe in Use for Metal Turning in 1880. Tools of Mushet Steel Doubled the Output of Such Machines

Courtesy of The Bettmann Archive

moving metal was an unsatisfactory and variable operation. It has often been said that the manufacture of carbide cutting tools is still in the laboratory stage. Nevertheless, with all the difficulties faced by the modern manufacturer—problems such as the control of tungsten grades and impurities and continuing experiments and discoveries that so often throw out the old ways and bring in new and strange ones almost overnight—his product is highly precise, predictable, and efficient when compared with that of the metalworker in the middle of the nineteenth century.

Mushet Steel. It was about 1850 that scientific experimentation began to find a better way to cut metal and to develop improved tools and materials with which to do it. However, the first of the so-called high-speed steels probably was the result of an accident. In 1868 Robert Mushet stumbled on what became known in England as Mushet steel. According to the story, experiments with the use of manganese as a ferrous alloy were being made at Mushet's foundry. In the course of the

work it was discovered that one of the bars of steel had the strange property of hardening in the air without quenching or cooling of any kind. Mushet found that this bar contained tungsten, which probably was there as an impurity.

Mushet, no doubt, was as much surprised at the results as the rest of the world was delighted. In any event, his discovery led to many experiments with various chemical elements in all varieties of combinations and proportions. As a result it was not long before an alloy steel was obtained which was much better for fast machining work than carbon steel. Fig. 3 shows a metal turning lathe of this period.

Eventually it was found that the quality of the steel could be much improved if the end of the bar to be used as the cutting edge were reheated and then cooled in an air blast instead of being allowed to cool by itself. Thus, in America the material became known as air-hardening steel. In the development that ensued, the tungsten was added in the presence of a much higher percentage of manganese than ordinarily used. Air-hardening, or so-called "self-hardened" steel, today has a composition of about 5.441 per cent tungsten, 0.398 chromium, 2.15 carbon, 1.578 manganese, and 1.044 silicon. The rest is, of course, iron.

In spite of the fact that such steel could withstand much higher temperatures than carbon steel, it was not until about 1890 that machine shops began its general adoption. It made possible the machining of cast iron at a rate of 48 f.p.m. and steel at a rate of 36 f.p.m., an increase of approximately 90 and 45 per cent respectively. It was about this time that Frederick Taylor, working in America, discovered that a still greater gain in cutting speed could be obtained if a stream of water were thrown on the chip or on the nose of the tool. Until Taylor's time, the makers of air-hardening steels are said to have warned every buyer never to use any water on their tools during cutting operations.

It was the experiments of Taylor and his associate, Maunsel White, which brought about the use of chromium in combination with tungsten and manganese, and paved the way for the present type of high-speed steel for machining. At the time Taylor began his experiments, machining speeds were still ranging in the neighborhood of 40 f.p.m. and the quality as "red hardness" was unknown. After Taylor's experiments, it was possible to keep "high-speed" tools in cutting condition even though they were operated at red heat. During the Paris Exposition of 1900, visitors were astonished to see a machine taking such heavy cuts and at such a high speed that the shavings were at a blue heat while the tool itself showed red hot. The development of these steels made possible the highly specialized machine tools for the automotive industry and for the construction of highly armored warships. For the first time it was feasible to machine huge castings as shown in Fig. 4.

At one time, Taylor estimated that he had cut up some 800,000 pounds of steel forgings and had made more than 40,000 tests. As a result of his patient and studious work, cutting speeds were increased nearly 25 per cent and, during the decade from 1900 to 1910, metal

cutting was done at what was then considered the marvelous speed of 75 f.p.m.

However, Taylor's greatest contribution to the advance of metal cutting, and consequently to the advancement of civilization in general, was not merely the increasing of the speed at which metal could be cut. His

Fig. 4. Tools of High-Speed Steel Were Shown at the Paris Exposition in 1900, Where Visitors Were Astounded To See Shavings Removed at "Blue" Heat by a Red-Hot Tool

Courtesy of The Bettmann Archive

experiments made it possible for such cutting to be done with tools that stayed in usable condition longer than ever before. With the old carbon-steel tools, resharpening was necessary at frequent intervals. If the tools were operated at a reasonably high rate, they were likely to collapse entirely. The new, high-speed, tungsten-chromium steel that Taylor developed, together with his method for heat-treating it, made great gains in tool life. These were the truly important phases of his discoveries.

The Cast Alloys. About 1915 an entirely new material was introduced to the field. This was a nonferrous alloy composed chiefly of cobalt, chromium, tungsten and certain other additions, depending upon

the manufacturer. One of the earliest producers named it Stellite. This name has become a somewhat generic term for all similar materials made today, regardless of their manufacturing source. This alloy is not appreciably affected by temperatures up to 1500° F. In fact, a Stellite tool cuts even better at a dull red heat than when cold. This material, which came to be known at a later date as a "cast alloy" as distinguished from the sintered carbides, not only permitted machining at greater rates of speed but again reduced the number of resharpenings required for any given job. A classic example of its machining ability is that it can make a 1/4" deep cut approximately two miles in length with one sharpening.

Stellite, Tantung, and the rest of these metals, whatever their trade names, are the cast alloys which will be considered in detail in later sections of the book. They are extremely hard, exceedingly brittle, and quite resistant to abrasion. But, because they are cast and can be machined only by certain sintered carbides, and then with difficulty, and because they are so exceptionally hard, they are difficult materials from which to make tools. The advances that resulted from the use of the Stellite type of alloy can perhaps best be shown by the fact that 50 years before its introduction it took three hours to perform an operation that can now be done in less than a single hour. This is approximately a 200 per cent increase in efficiency even over the high-speed tool steel developed by Frederick Taylor.

Thus, the beginning of World War I foreshadowed the revolution that came to full fruition in World War II. It gave impetus to the American way of life—the enjoyment of a higher standard of living. Cheaper production of goods in greater quantity meant that more people could buy them.

Super-High-Speed Steel. The next forward step occurred in 1928 when a material popularly called "super-high-speed" steel was developed by adding cobalt to the high-speed steel of a century before. This made it possible to machine at a rate as high as 200 f.p.m. which meant that ten parts could now be produced where only eight had been made previously. Super-high-speed steel was particularly useful for the machining of materials such as steel, cast iron and the ferrous alloys, and the then new industrial metals such as aluminum and magnesium.

It was at about this same time that powder metallurgy presented the tool industry with a revolutionary method and material which developed with special rapidity during the wartime years. This was the sintered carbide material with which this book is particularly concerned.

Perhaps it would be well at this point to summarize briefly the industrial progress and improved standards of living which resulted from the milestones in cutting tool history up to 1928:

In 1800, the only tool material available was plain, high-carbon steel, which was capable of cutting from 20 to 40 f.p.m. at a temperature under 400° F. In 1850, the only materials in use were high-carbon steels and the alloy carbon tool steels, which were capable of cutting

from 30 to 40 f.p.m. but still at a temperature no higher than 400° F.

In 1868 came the development of the semi-high-speed tool steels by Robert Mushet, capable of cutting from 40 to 60 f.p.m. at a maximum tool temperature of 500° F. High-speed tool steel, such as that developed in America by Taylor and White, was first used extensively in 1900. This was actually not a new alloy but rather a radical heat-treatment of tool steels which gave them the property of red hardness.

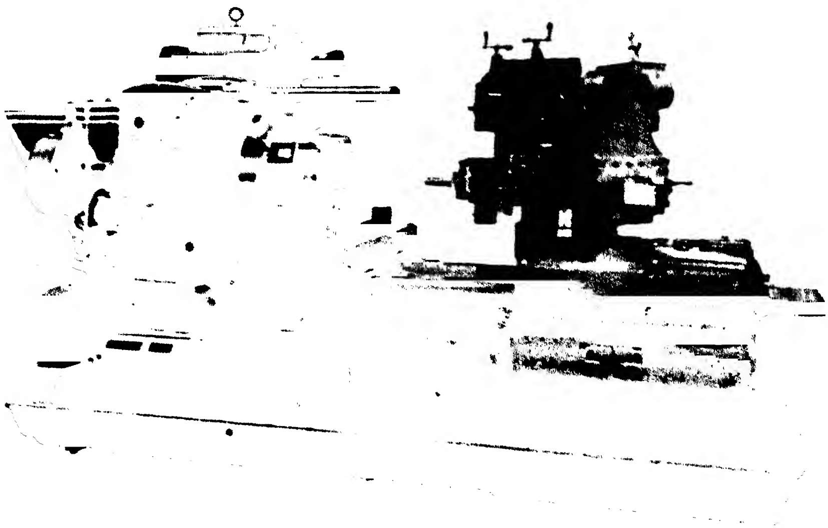


Fig. 5. Single-Spindle, Chucking, Saddle-Type Turret Lathe
Courtesy of Potter & Johnson Machine Co

These steels could cut from 50 to 75 f.p.m. and at the then remarkable temperature of 1100° F. The introduction of the cast alloys—Stellite, Tantung, and others, capable of cutting from 80 to 100 f.p.m. at 1500° F.—came in 1915. Finally, in 1928 came the almost simultaneous development of super-high-speed steel and the sintered, tungsten-base carbides. The super-high-speed steels were tool steels to which cobalt had been added, making possible the cutting of steel at 65 to 100 f.p.m. at temperatures up to 1600° F. The carbides made possible the cutting of ferrous metals at the rate of 150 to 500 f.p.m. or more at temperatures up to 2000° F.

The Carbide Era. The sintered carbides are the result of a startling new technique in metallurgy which, without exaggeration, has changed much of American industry. Through ingenious use of these materials in the cutting tools of machines such as the turret lathe shown in Fig. 5, the country has been given almost unbelievable production capacity.

The first rough outlines of the new methods of powder metallurgy

were drawn in 1909 when W. D. Coolidge of the General Electric Company discovered a method for the production of tungsten wire for lamp filaments. It had long been believed that tungsten would be ideal for this purpose because of its ability to withstand terrific temperatures. Tungsten melts at the highest temperature of any of the metals. Only carbon, among the elements, has a higher melting point. Since it would be difficult to produce a furnace which will fuse tungsten without itself burning up, it is produced in its pure form by chemical processes.

Coolidge's problem was to find some method of getting the tungsten into wire form. His process began by dividing the metal into a fine powder. He then compressed, sintered, and heat-treated it at temperatures far below its melting point, forming it into ingots which could be drawn into wire finer than one thousandth of an inch in diameter and with a tensile strength[1] of approximately 600,000 pounds per square inch. It was thus that Coolidge, perhaps with little or no thought of its scope, presented metallurgy with a new concept: that cohesion could be had without fusion (melting). This is the basis of the whole powder metallurgy technique.

Sintering, for those unfamiliar with the process, is defined as "cohesion of the particles in the compound." It may be accomplished, and often is, at room temperatures. Tungsten and its carbides, however, are sintered at temperatures ranging from 2500° to 2660° F., depending on the process and the use to be made of the sintered mass.

Oddly enough, World War I almost prevented the development of this discovery in America. Germany is believed to have used the technique and actually to have made tungsten carbide during World War I. In any event, immediately after the war the product, which was first known as "Widia," was introduced to the rest of the world, and its use in the Krupp works for the production of big guns and other war materials revealed.

The first tungsten carbide for cutting was shown to American industry as a laboratory material early in the Twenties. In those days it was a common stunt to place a milk bottle in a lathe and in a few seconds cut off the neck with a tungsten-carbide cutting tool. Its action in cast iron, which had long presented industry with one of its most difficult machining problems, was equally remarkable.

Through the use of tungsten carbides, surface foot rates were increased more in one sudden leap than they had advanced in all previous industrial history. It was not at all unusual to cut up to 300 f.p.m. using tungsten-carbide tools. Thus, in less than 25 years, surface foot rates jumped from 75 to 300 feet or more. Since the material for cutting tools was far better than most of the machines that could use it, a feverish development of machine tools took place in an effort to keep up with the advance of the cutting tools.

[1]Tensile strength is the resistance of a material to a force which seeks to pull it apart. It is usually given in pounds per square inch or p.s.i.

TABLE I. COMPARISON OF VARIOUS HARDNESS SCALES

Moh's Scale	Mineral	Carborundum Number	Brinell or Indentation Scale	Rockwell C Number
1	Talc	1		
2	Gypsum	2	32	
3	Calcite	3	135	
4	Fluorite	4	163	
5	Apatite	5	360-430	39-45
6	Feldspar	6	560	56
7	Quartz	7	710-790	61-64
8	Topaz	8	1250	72
9	Sapphire	9	1445	75
	Fused alumina	10	1635	77
	Tungsten carbide	12	1850	80
	Silicon carbide	14	2150	84
	Boron carbide	19.7	2250	85
	Diamond			
	Carbonado	36.4	8200	
	Congo (gray)	37.8	8275	
	Congo (yellow)	41.0	8450	
10	Brazil ballas	42.0	8500	

NOTE.—The original Moh's scale consisted only of the relative hardness values given in the first column. Carborundum, Brinell, and Rockwell C scales are shown for comparison.

While tungsten carbide, as it was originally introduced, had remarkable cutting characteristics on cast iron, it did not prove too successful in the machining of steel and its alloys. It was possible, on the other hand, to machine aluminum and its alloys and most of the other light metals at rates as high as 1,500 feet or more a minute. But, because of tungsten carbide's inability to do more than a fair job on steel, an intensive laboratory search began for other or similar materials which would do everything with steel and the ferrous alloys that tungsten carbide was capable of doing with cast iron and the light metals. It is largely in this field of research that the tremendous development of the last three or four years has taken place.

A comparison of the relative hardness of the various compounds from which cutting tools are made is necessary in order to appreciate this development. Table I presents Moh's relative hardness scale as well as parts of other hardness scales for the purpose of collation. In addition, the proper interpretation of terms as used in this book should be understood. For example, metal is any of the chemical elements commonly considered as such, together with any or all of its alloys.

Carbon, boron, and silicon are not classed as metals but rather as metalloids. The boron carbides, for instance, under this classification, would be considered the hardest man-made materials but not the hardest man-made metals. Additional material on the nonmetallic carbides will be presented in subsequent chapters. Next to the diamond, then, boron carbide is the hardest material known to man, and tungsten carbide is the hardest metal known to man.

Tungsten carbide is made in modern factories by powdering the tungsten and adding powdered carbon. This combination is then mixed with a given percentage of cobalt and is milled for four to six days with tungsten carbide balls so as to reduce the size of the powder and coat each particle of carbide powder with a thin film of cobalt. Following this, the powder thus prepared is compressed by dies into tablets or blanks of the desired shape under pressure of from two to 20 tons per square inch. The "green" pressed blocks or shapes are placed in a high-reducing-atmosphere furnace until the blocks become a compact and, in most cases, a dense mass. Once formed and heated, the metal cannot be poured, cast, or heat-treated. However, it can be cut with a diamond wheel and ground with a number of abrasives into almost any shape, sharpness, or finish desired.

Tungsten carbide is generally used in quite small pieces, a fact which is often wrongly attributed to high cost. While it is true that the material cost in the neighborhood of $440 a pound at the time of its introduction, tungsten carbide today is selling at about $16 a pound. However, it is never sold by the pound. Instead, the kilogram is used. The tungsten carbide business is one of the few American industries in which the metric system of weights and measures prevails. This is probably because the micron is used as the measure of particle size in practically all metals powders. The micron is 1/1,000 of a millimeter or about 40 millionths of an inch.

These small pieces of tungsten carbide are brazed to steel or alloy tool shanks to form tips, blades, edges, etc., for two very good reasons. First, the steel provides the necessary backing and support for the tungsten carbide which is quite brittle and low in tensile strength. Second, tungsten carbide itself cannot easily be formed into the myriad shapes for bits, milling cutters, drills, reamers, shapers, boring bars, inside broaches, and the other tools demanded by modern industry.

First Carbide Improvement. The first variation of tungsten carbide was a tungsten-tantalum carbide mixture first shown to industry in 1931 by the Fansteel Metallurgical Corporation. Similar to plain tungsten carbide, it also was a product of powder metallurgy. The tantalum (or sometimes columbium) and tungsten powders are added to powdered carbon and heated to form the carbide. Then, in the usual process, the carbide is repowdered by ball milling, during which it is mixed and coated with cobalt. In the sintering process, the cobalt or such other binders (sometimes nickel) as may be used to form the matrix, goes into a more or less plastic state, cementing together the

Fig 6A *(Top)* Vertical Face Milling of S A E 4340 (Chrome Moly Nickel) Steel without Coolant

Fig 6B *(Bottom)* Head of Turret Type Machine with Special Features for Boring, Facing, and Chamfering Cylinder Pads on a Magnesium Aircraft Engine Crankcase

crystals of tungsten carbide, tantalum carbide, and any free crystals of other metals present. The whole process is known to metallurgists as the "network phenomenon." However, it is not the intent of this book to delve further into these matters. Additional information can be found in the many textbooks on metallurgy and physical chemistry.

The tungsten-tantalum carbide combinations proved highly successful in the machining of steel at much higher rates than had previously been possible. These rates were as much as 150% greater in such machine tools as lathes, planers, shapers, boring mills, and others using single-pointed bits.

While this increase in the work rate was definitely an improvement, there were still many limitations. It was found that this material afforded little advantage over tungsten carbide in multiple tool applications similar to those shown in (A) and (B) of Fig. 6. Neither tungsten carbide, tantalum carbide, nor their combinations appeared to have the ability to stay on the job at very high speeds in a cutter having many edges or blades. Eventually it was discovered that part of this difficulty was the fault of the tool designers and not the material. Fewer teeth in the tool solved most of the problem.

The milling machine, with its discontinuous cutting action, has long been industry's problem child. Any cutting material used in it is subjected to terrific strains as well as to considerable heat. In spite of this, it is one of the most widely used machines in modern industry. It was essential, therefore, that a tool be designed which would permit successful application of the carbides to milling operations.

The Second Variation. Many metallurgists have sought or taken credit for the next "discovery," tungsten-titanium carbide, which is now being manufactured by perhaps a dozen widely known organizations. There have been almost as many different versions of the method used in the manufacture of tungsten-titanium carbide as there are producers of it. There have also been almost as many assertions of theory as to the exact form and composition of the material. However, these are matters which need not be decided here.

One manufacturer says he produces tungsten-titanium carbide in a menstruum, or bath of molten nickel, then reduces this with acid to a double carbide of tungsten and titanium with the presumed formula $WTiC_2$. This process seems long, arduous, and costly. It could hardly be said to be an efficient commercial method for manufacturing the material. Whether the resulting compound is actually a double carbide is also a matter on which metallurgists do not agree. Most of them hold that it is a eutectic alloy[2] or a mixture of monocarbides and not a true chemical compound.

[2]Explaining the term "eutectic alloy" requires first, a workable definition of an alloy. Without going into too much detail, an alloy is a "solid solution" of various metals. And, just as all salts are not soluble in water, neither will any metal combine or alloy with every other.

To carry the salt analogy further, it will be recalled that water at normal tem-

This manufacturer states he then powders the $WTiC_2$ crystals he has thus produced, mixes them with powdered tungsten carbide in the desired proportions, adds his binder, forms the powders into size and shape in a die, and sinters the mass.

This again seems a highly impractical method of accomplishing an end which is commonly done by others simply by mixing WC and TiC powders, adding cobalt during the ball milling, compressing the whole into compacts, and sintering.

It should be said that tungsten-titanium carbide, whether an alloy, a mixture, or a true double carbide, has a higher melting point, higher hardness number, and greater break strength than any of its components either as metals or carbides. Although the proportion of titanium carbide in these cutting tools is quite small, it is extremely potent in its effects.

Many other combinations of these metals and alloys have followed. Quite often, compounds of tantalum, titanium, and tungsten carbides are used for one cutting purpose or another. Unlike titanium, the proportion of tantalum may range all the way up to pure tantalum carbide with a binder, usually cobalt, but which, for special purposes, may be nickel.

The introduction of these variants to tungsten carbide, beginning about 1938, in conjunction a little later with certain tool improvements and a new method of using them known as "negative rake," turned out to be the answer to many machine tool users' prayers. This was especially true in the case of milling machines. These developments will be treated in detail in later chapters of the book.

Negative Rake. While this later study of positive and negative rake angles will consider not only milling cutters and other multiple-pointed tools but single-pointed applications as well, it is appropriate at this time to give, in broad outline, the differences involved. Until about 1943, it was standard practice to use any cutting tool, whether in a single- or multiple-point installation, in what theoreticians called the positive rake position. As shown in (A) of Fig. 7, this meant that the

perature will dissolve table salt up to a certain point, forming what is then known as a "saturated solution." If such a term were not peculiar to the fields of metallurgy and physical chemistry, the saturated solution might just as well be called a "eutectic solution."

However, water, when heated, takes up additional salt into solution and, as it cools, precipitates or drops the excess over the saturation point. Metal alloys do the same thing. The eutectic alloy, then, is the mixture (which may and often does include actual chemical compounds) in which the proportions of the ingredients are ideal to form a solid solution. The eutectic point is the temperature at which, during the cooling of the mixture, the excess of any element present is precipitated; or, on heating, excesses of elements or compounds in the solution are taken up.

This brief explanation does not by any means cover all the phenomena of the process. To do a thorough job would take a shelf of books. As a matter of fact, a tremendous number of books are available on steel alone, our best known and commonest alloy. Since such a comprehensive study is impossible in this book, additional information, if it is desired, should be obtained from any of the reference texts on the subject.

tool was pointed at the work much as one would use a cold chisel to shear off unwanted material. In negative rake cutting, however, the tool is pointed not against the movement of the work, but in the same direction as illustrated at (B). This means that the tool, to use a simple analogy, is employed more in the fashion of a scraper. The action of the scraper is shown at (C) in Fig. 7.

While positive rake cutting causes the material to be sheared off, negative rake cutting causes it to be crushed off. The most pronounced difference lies in the fact that a much thicker chip is removed by nega-

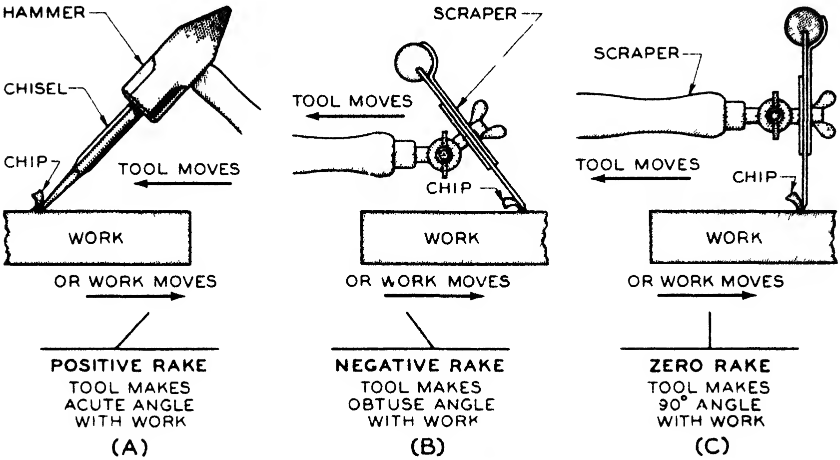


Fig. 7. Sketches Illustrating in Simple Form the Difference between Positive, Zero, and Negative Rake

tive rake cutting. There are other considerations, chiefly the matter of power and the ability of the machine itself to withstand greatly added loads. These factors will be considered in latter portions of the book.

The results of negative rake cutting with the newer and harder tantalum- and titanium-tungsten carbides proved almost fantastic. In positive rake cutting, the rate at which a cutting edge was revolved in a high-speed steel cutter (that is, the surface foot rate, or f.p.m.) seldom was more than 80. The range normally was from 60 to 80 and almost never higher than 90. But with negative rake angles and carbide-tipped cutters, the surface foot rate range may be anywhere from 600 f.p.m. up, especially in machining the light metals. Many operations, even on soft steel, are being performed at this time at rates considerably over 2,000 f.p.m. In fact, in one instance a manufacturer is using a speed of 3,500 f.p.m.

As will be seen later, another advantage of the new methods and materials is that fewer cutting edges are required, with a consequent greater load per tool bit. This simplifies the design of cutters in many instances, and reduces cost.

Not to discount some of the increases which have been revealed, it should be said that the average gain on steel is around ten to fifteen times, or between 1,000 and 1,500 per cent, a figure which is sufficiently astounding to place it well within the history of the development of the carbide tools.

It will be seen, nevertheless, that where a milling cutter in the old days—and "old" here is anything from five to ten years back—was traditionally set to run at about 60 f.p.m. It may now run as high as 15 times that rate. An increase has been made likewise in the rate at which work is fed into the cutter. From a few inches per minute, this rate has been stepped up to as much as 50 and 60 inches per minute, provided the power is available. For instance, the milling of such material as S.A.E. 4340[3] chrome molybdenum steel now may be done at 35 inches a minute in machines built and powered to take such rates. This material formerly was cut at from four to six inches a minute, which was considered fast milling.

It must be remembered that low cutting rates are still being widely used by manufacturers who have not yet converted to the use of carbide tools, either because they couldn't get the machines or because of lack of

[3]The S.A.E. (Society of Automotive Engineers) system of designating steels constitutes a code for the easy recognition of the approximate makeup of most steel alloys. Usually, the first digit in the designation tells what principal element, other than iron, is used in the alloy, although the first two digits, and sometimes the first three are required, as will be explained. In general, the first figure of the code number means: (1) carbon, (2) nickel, (4) molybdenum, (5) chromium, (7) tungsten. The (3) designation is reserved for nickel-chrome steels, (6) for chrome-vanadium, and (9) for silicon-manganese. This does not mean, however, that only those elements will be found in any of the steels so designated. It is merely a convenient system of grouping alloys.

For example, the 10xx series in the S.A.E. system includes all the plain carbon steels with varying percentages of manganese. The 11xx and 13xx series of carbon steels are the "free-cutting" types. They also include manganese, but with more sulphur and phosphorus than the plain types. The T13xx series is the manganese tool steels. All the series beginning with 20 to 25 are nickel steels, again with manganese. The 3xxx series is nickel-chromium-manganese steels. The 4xxx is molybdenum steels with manganese. Numbers 41 to 43 are molybdenum steels with added chromium. Numbers 43 to 48 are molybdenum steels with added nickel. The 5xxx series is all chromium steels with manganese; the 6xxx, chrome-vanadium with manganese; the 7xxx, tungsten with chromium and manganese; the 8xxx, molybdenum with which may be combined manganese, nickel, and/or chromium; and the 9xxx, silicon and manganese.

It should be noted in particular that the carbon content, shown by the last two or three digits in the designation, is expressed in hundredths of a per cent. That is, a plain carbon steel such as 1025 would have a carbon content ranging from 0.20 to 0.30, or an average of 0.25; while a 52100 steel would be a chromium steel with a carbon content ranging from about 0.95 to 1.05 per cent, or an average range of 1.00 per cent.

An X in front of the numerical designation indicates a free-cutting type steel; a T, a tool steel; and the three number designations such as the 309xx, the 512xx to 517xx, and the 713xx and 716xx series, denote that the alloy is corrosion resistant, heat resistant, or both.

sufficient power to operate them. Actually, few changes in production equipment are needed to tool up for the use of carbides. While this point will be covered more extensively in the next chapter, it may be said in general that what was formerly done on a five hp machine can

Fig. 8 The Use of a Coolant in High-Speed Operation Is Most Important

be done with carbide and negative rake on a 15 hp machine without any radical change except to slip a carbide tool into the tool post or the milling cutter arbor. However, to achieve the radically high speed and feed records currently being set is another story which will be enlarged upon subsequently.

The Coolant Controversy. When metal is cut in a turret lathe, a coolant is always used as shown in Fig. 8. The use of a coolant is absolutely necessary because of the high heats generated which tend either to break down the tool or to distort the work. High temperature at the tool point lowers the strength of the braze which holds the carbide tip to the tool shank. Again, while the carbide itself is not affected by temperature normally encountered in metal cutting, temperature does contribute materially to tool wear and cratering, and for this reason should be kept to a minimum. This may be accomplished by using a coolant.

In lathe practice, some operators assert that coolants, particularly

those containing sulphur, are detrimental to the carbide tools. This claim is not substantiated by results obtained in the industries nor in research laboratories. There is some evidence that sulphur, when suspended in cooling oil, reacts with cobalt (the carbide binding material), but the amount of reaction is usually negligible. Other operators claim that with negative rake carbide cutting tools, no coolant is necessary. This may be true, but only in cases where cutting operations are carried on at reduced speed.

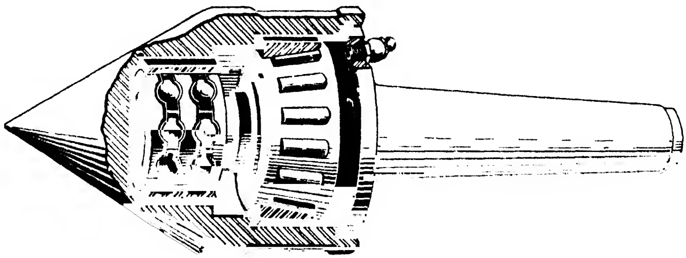


Fig. 9. The "Live" Center for Use on Lathes Where Lengthening of the Workpiece Due to Expansion Is a Problem
Courtesy of Marvel Tool and Machine Co

The use of a coolant permits higher speeds for the same tool life. A 25 per cent increase in speed usually may be anticipated when a coolant is used, regardless of positive or negative tool angles. Although cutting dry eliminates a messy and sloppy operation which has always been detested by the operators, the use of carbide tools demands that a copious quantity of coolant be continuously played on the cutting tool. In addition, cutting pressures with negative rake tools are greater than with positive rake. There is a greater demand for power and consequently more heat is generated which calls for the use of efficient coolants.

Problems of High-Speed Operation. High speeds on all types of machine tools have resulted in several problems not known to the early operator. On lathes, for example, there is the matter of centers. When carbide tools began to be used, carbide centers for lathes promptly followed. Because of the terrific heats generated and the fact that the cutting could continue to be done at high heat, the work often lengthened, causing binding against the center. This difficulty has been overcome to a large extent by the introduction of "live" centers, an example of which is shown in Fig. 9.

Carbide Tools and Iron and Steel. When the steel-cutting grades of carbide were introduced to American industry, the dramatic increase in cutting rates was particularly noticeable on milling machines. Since milling is one of the most widespread and essential of all machining operations in industry today, perhaps it would be well at

this point to look back a little at the history of milling in America.

Eli Whitney, with whom most people have become acquainted either as the inventor of the cotton gin or as the first man to standardize measurement within a factory, was also the inventor, in 1817, of the

Fig. 10. Eli Whitney and His Mass-Produced Rifle
Courtesy of DoAll Company

milling machine. It was the milling machine as much as his standardization of measurements that made possible the system of interchangeable parts which is the backbone of all modern mass production.

Up to Whitney's time, such materials as military rifles were made by craftsmen, each of whom finished a complete gun and assembled it himself. The parts from any one rifle would not necessarily fit any other. But in one of American history's most significant moments, Whitney conceived an idea and proposed it to Thomas Jefferson, then President of the United States. His suggestion was that a machine be built with which guns, having interchangeable parts, might be manufactured for the army on a mass production basis. Jefferson had foresight enough to see the possibilities of what was considered by most folk of the time a silly scheme, and arranged to have Whitney's plan financed. By standardizing the inch within his own plant and introducing the division of labor, Whitney was able for the first time in history to make a product by assembly line methods. In his plant, one man made the same part day after day, another worker made a second part, and so on, just

as is done in a modern shop. These parts were then brought to a central point for final assembly. Any of Whitney's rifles could be repaired by replacing the broken or defective part. Visiting a gunsmith or discarding the whole gun was no longer necessary.

The first assembly of a completed rifle was a dramatic occasion. When Whitney had built enough parts to assemble 100 guns, he brought in Jefferson and his advisors. One of the ordnance aides in the party was then shown the various disassembled parts of which the finished rifle was built. The group watched in amazement as the aide selected duplicate parts at random and assembled a complete rifle. An artist's conception of Whitney and his rifle with its interchangeable parts is shown in Fig. 10.

Whitney's first milling machine was powered by less than 1/4 hp and, of course, was equipped with carbon tool-steel cutters. During the time that was once required to produce one rifle, modern milling machines using the steel-cutting grades of carbide now could produce 1,550 completed firearms of the same 1817 design.

Machining the "Nonmachinable." Not only are carbide-equipped machines capable of turning out products and parts at much higher rates than were ever before known in industry, but one of the most significant characteristics of such tools is their ability to machine materials of a hardness believed impossible even a short time ago. For example, certain manganese alloys of great military value and post-war industrial importance, until very recently, could be cast only. With the aid of carbide tools, they can now be machined. An outstanding example is an injector body, a part subject to extremes of heat and great wear, which can now be machined at the rate of 170 f.p.m. Added to this, it is possible to make 40 pieces between grinds of the cutters, a figure that only a short time ago was considered good cutter life for materials which were easily machined.

Another instance which may be cited from wartime experience is the rough turning of 75 mm. shells at a speed of 196 f.p.m. In 40 seconds, 2 3/4 pounds of metal were removed and tool life was stretched out to 100 drop-forged shells between grindings. During World War I, a similar operation would have required 20 minutes.

It can be seen from this that the use of carbides has increased some kinds of production more than 2,000 per cent. What effect this will have on present operations in the manufacture of such products as automobiles, vacuum cleaners, refrigerators, and similar consumer goods, can only be a guess.

Significant cost reduction is found not only in the manufacture of finished parts but also in the steel mills where the price of a part or product begins. The rolling of slabs, bars, plates, and shapes is often the first step in the making of any metal product. Until a short while back, rolls for these mills were machined with high-speed steel in approximately 13 hours. Today these same rolls are turned in a little over four hours, a saving, roughly, of 66 per cent.

The many problems that arose in dealing with armor plate were a bottleneck in military production during the last two wars. Armor plate is exceedingly hard, and, until a short while before World War II ended, was placed in the realm of nonmachinable material. It is a cast alloy steel having a hardness of 400 and up on the Brinell scale.[4] It usually contains "hard spots" and sand pockets, and the best machining speed before the introduction of the carbides was 11 f.p.m. Even at that rate, the high-speed tool usually broke down after some eight inches of wear. Carbide tools machined this material at 60 f.p.m. and did 31 inches between grinds, a 500 per cent increase in production and an increase of about 300 per cent in the time the tool stayed on the machine. This last was an important factor because the increased time of the tool on the machine meant greatly added production. There was less "down" time and less time lost regrinding and retooling.

Gear blanks, used on milling machines, are now being machined with steel-cutting carbide at 500 to 600 f.p.m. with some 150 pieces being processed between grinds. In the old days six pieces was the normal number for high-speed cutting tools. This is a 400 per cent increase in production and a 2,500 per cent increase in tool life. With high-speed tool steel, a certain vital airplane part of chrome-molybdenum steel formerly was milled at 3 f.p.m. Carbide milling cutters perform this same operation at 632 f.p.m.--a 2,000 per cent increase.

Chrome-molybdenum bars could be sawed with a milling cutter at a rate of 1 3/8 inches per minute. A carbide milling saw does the operation at 20 inches per minute. The surface foot rate under the old method was 83; under the new it is 833, an increase of 1,000 per cent.

An increase of 400 per cent is shown in the case of another vital aircraft part, now being machined in less than half a minute with carbide cutters set for negative rake. This same part formerly required 2 1/2 minutes for the same operation. In addition to the saving in time and tool life, the new technique requires no coolants, makes possible the machining of steel parts with one operation instead of two, and turns them out with a finish which makes a subsequent grinding operation unnecessary.

These are only a few instances given here to show the vast possibilities that are now being explored for the use of such tools. The upper limits are not known to anyone. Every day, reports come in of operations being done at almost unbelievable speeds. Only recently it was revealed that an aircraft part of tough chrome-molybdenum steel is now being milled at a rate of 2,500 f.p.m., an almost astronomical figure which only a short while back was considered possible only in the realm of the lighter and softer metals.

Needless to say, all these tremendous increases in the speed of milling have thrown a great many problems in the laps of tool designers, shop foremen, tool buyers, tool engineers, and the rest of management.

---

[4] The Brinell scale is one of several in common use for measuring relative hardness. Others often used are the Rockwell "A," Rockwell "C," and scleroscope.

Fig. 11. Milling S.A.E. 4340 Steel on a Duplex Milling Machine
Courtesy of Kearney & Trecker Corp

To keep up with the speeds possible with the new cutting materials, it has often been necessary to redesign, if not whole machines, at least many of their parts such as bearings, spindles, arbors, knees, and beds.

One of the outstanding features of the use of carbide tools is the fine finish obtained. An aircraft wing hinge (the part which couples the wings of an airplane to the air frame), for example, must be finished with an almost mirror-like surface, free from even the slightest blemish. Through the use of carbide tools, this surface is achieved directly in the milling operation without any subsequent grinding, lapping, or polishing. The material ranges in hardness from 200 to 400 on the Brinell scale, depending on specification. With old tools, it was possible in the hardest grades to cut about 40 f.p.m. on the roughing cut only. With the new tools, the surface foot rate is 564, while the same effect as with rough, finish, grind, and polish is obtained in a single operation. Fig. 11 illustrates the operation as it is performed.

Carbide Tools and Light Metals and Plastics. The sintered carbides were first introduced more than 20 years ago as a cutting material for aluminum and its alloys, bronze, brass, magnesium, and the nonmetallic plastics. They were highly successful, but, as with

iron and steel, it has been only in the last few years that any real progress has been made in the development of the high-speed machining of these materials. The outlook regarding the upper limits of feeds and speeds for these materials is similar to the situation concerning the machining of iron and steel. No one knows what the possibilities are. The only limiting factor at this time is the design of the machine tools themselves.

As was pointed out in the preceding paragraphs on the machining of iron and steel, such factors as spindle bearings on milling machines and centers on lathes have been the seat of the problem caused by the high speeds suddenly coming into practice. Spindle bearings thus far have not been designed to give satisfactory service at speeds in excess of 10,000 r.p.m.—at least not on machines where there is sufficient force exerted to remove the large amounts of metal usually accompanying negative rake cutting. But even at the limit of 10,000 r.p.m., production has been pushed up 1,000 per cent or more, which means that ten pieces are now being machined in the same time in which one was produced only a short while back.

To take up the light metals in some sort of order, it is best to begin with the very hard, nonferrous materials which are normally regarded as being quite resistant to machining of any sort. First among these probably would be aluminum bronze. A typical example of the problem included is the turning of a ring casting on a boring mill. Up to the time of the most recent high-speed developments, this casting was machined at a speed of around 60 f.p.m. Today it is being machined at 200 f.p.m., a 5/16" cut being removed at a feed of .001". All this adds up to a total production increase of almost 300 per cent.

Another case is that of a manganese bronze part which is now being milled in one-tenth the time formerly required by high-speed cutters. To be specific, the present rate is 820 f.p.m. with a feed of 20 inches. Machine time is now eight seconds, whereas a short time back this same operation took 90 seconds.

An aluminum radio part which is now being straddle milled at 2,000 f.p.m. at a feed of 20 inches, is another good example. This part was formerly cut with high-speed tools at a feed rate of seven inches. Also, 4,800 pieces are now being cut between grinds, where previously, only 100 pieces could be milled before the cutters needed resharpening.

An aluminum casting is now being milled at 3,400 f.p.m. and at a feed rate of 75 inches a minute. Until a short time ago, ten of these castings milled in an hour was considered good production. By present standards, 125 an hour is ordinary. As in the case of the aircraft wing hinge, surface finish is of great importance in this particular operation. Again, since no coolant is necessary, the total overall increase in efficiency is even greater than the 12.5 times indicated by the milling rate.

All of these startling increases are not limited to the milling machine. There is the case of a planing operation on duralumin, an alloy

of aluminum and copper. More than 200 cubic inches of this metal are being removed on such a machine per minute. The speed of the cutter is 5,250 f.p.m. while the feed is 22 inches per minute. Those who have long been familiar with the operation of machine tools will find this all the more remarkable when it is stated that the cut in this particular instance has dimensions 2 1/2'' wide, 3'' deep, and 96'' long.

Such parts as dural sheets are being scarfed (beveled) at 5,470 f.p.m. while the work is being fed into the cutter at the incredible rate of 150 inches a minute. A small, 2 1/2'' cutter is performing this operation in a West Coast aircraft plant. In this instance, spindle speed is 8,000 r.p.m., the equivalent of 5,470 f.p.m. This is similar to another instance reported from the West Coast where a part of the same material is being machined at 5,070 f.p.m. at a cutter speed of 8,500 r.p.m. The feed rate is 100 inches a minute. Until carbide cutters were installed, the maximum surface foot rate for this material was 1,500. Thus, more than a three-time increase in production is shown.

At least some of the credit for these startling gains should be given to an engineer named Art Schwartz who, until 1939, had been engaged mostly in woodworking. He transferred his activities to the aircraft industry at the outbreak of the war, and, with the characteristic enthusiasm and assurance of the uninformed, attempted to cut metal at the much higher speeds with which wood is worked. He made experiments that probably would have raised the hair of an orthodox designer, had one been present. He rebuilt machine tools, stepped up spindle speeds, changed some of the angles in the carbide cutters, and made alterations in the manner of putting the tips on the tools. But the one change that did have unquestioned value was his reduction in the number of teeth per cutter and consequent increase in the load per tooth. By these means, he pushed up speeds from 1,500 to as high as 20,000 f.p.m. and began milling many aircraft parts of small cross section at 4000 to 10,000 f.p.m.

In some cases, perhaps as the result of Schwartz's experiments, many parts similar in size and shape but composed of the hard, tough dural of the 14 and 24 ST series were being milled in his and other plants when the war ended at rates as high as 20,000 f.p.m. Neither Schwartz nor his confreres are prepared yet to say that this is the top limit. They just do not know.

Another field into which carbide tools have come of late is the tapering of asbestos lining for brake drums. Here, an increase in production of more than ten times that of a few months previous is being shown. In addition, the carbide cutters withstand the abrasive action of this material a much longer time than any other type of tool.

What once was considered a rate possible only in the turning and shaping of wood is now being used in magnesium. As with aluminum and its alloys, no one in authority is yet ready to say what the upper extremes of metal removal in this field may be. It appears that the only limiting factor is the design of spindles and spindle bearings. It

may well be that when machine design has caught up with the tools available, speeds two or three times those now being used may be common practice. In the case of magnesium, an interesting point is that so long as negative rakes are used and large chips removed, the material may be machined at these terrific cutting rates without fear of the metal suddenly flaming up. When magnesium is machined in small chips producing dust which is exposed to the air, fire is always a serious hazard.

In the plastics field there are even fewer facts available than are known concerning magnesium, aluminum, and their alloys. Generally, it may be said that the plastics are more easily machined than either aluminum or magnesium. What rates eventually will be determined for these materials is almost entirely a matter of guess. Furthermore, the plastics themselves are constantly being changed and new materials are being developed for specific purposes, each with its special characteristics, so that it will be years, perhaps, before any definite recommendations can be made for them. However, it is safe to say that at least a ten time increase in the speed of machining these materials is easily attainable with the use of carbide tools. It can also be confidently predicted that these rates will continue to rise.

Most of the foregoing examples cited have been of war materials, but it is evident at once that the advances and increases which have resulted from wartime production can be and have been carried over into the postwar and peacetime future with consequent lower costs, greater savings, and greatly increased production of almost everything America will want.

Machine Design Problems. It might be well at this point to discuss in greater detail some of the matters mentioned regarding spindle and bearing design which have delayed the even greater increases believed possible in machine speeds. First of all, it should be noted that while current spindle speeds of 10,000 r.p.m. or surface foot rates as high as 20,000 f.p.m. have been reached (actually with an 8" cutter and a spindle speed of 10,000 r.p.m., the surface foot rate per minute is considerably more than 20,000) the industry is still crying for higher speeds.

When carbides were applied to the milling of hard aluminum alloys, the first specially designed heads were built for a spindle speed of 15,000 r.p.m. When negative rake cutting came in with its attendant high torques and the work which normally would be done on a five hp machine was transferred to one of 15 hp, bearing troubles became pronounced. It was then that most shops limited their speeds to a maximum of 10,000 r.p.m. But, even at this speed it was necessary to control bearing temperatures by means of water cooling.

High spindle speeds alone cause no difficulty. They have been used for generations in the woodworking industry. But when high speed is combined with a high torque and a great increase in load, overheating and seizure difficulties appear. Most machine tool designers feel that

with the heavy cuts and the higher speeds which are demanded and will be increasingly insisted upon in the future, something new in the approach to bearing design and manufacture may be called for. Just what trend this may take is impossible to predict at this time. However, it can be said that the tungsten-base carbides, so well known and widely used now as cutting materials throughout all industry, may be the answer to the search for adequate bearing materials. Such work as has been done, and much of it was delayed because of wartime demands, indicates that these materials are suitable for bearing purposes. There are other hard carbides which may prove to be equally suitable, among them boron carbide, particularly the B_2C form, a nonmetallic material which has been mentioned before. The B_2C boron carbide (having more carbon than the B_4C, or abrasive form) tends to be somewhat self-lubricating. The friction coefficient is much lower than that of other materials with known ability to carry the load. B_2C's other characteristics, such as hardness and heat conductivity, indicate, from a purely theoretical standpoint, that it may make ideal bearing material.

A number of organizations have tests of all sorts under way. Results of these tests may prove of great and startling value. Some surprising results have come about through the inclusion in bearing alloys of a little known metal, indium. It has had remarkable effects and may prove to be the answer. There is also a new process of tungsten plating, kept under military wraps for several years, that may be the solution not only in bearings, but in many other industrial problems as well.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Who is the man generally credited with introducing the period of power-driven tools?
2. When was the first planer perfected, and what was the cost rate per job that was established?
3. What was the tool material used on early planers, and why were only the lowest speeds practical?
4. What was "Mushet Steel?" How was it developed?
5. What is air-hardening steel? What developments have resulted from its discovery?
6. Who is responsible for the first attempt at the use of a coolant, and what other achievements are attributed to this man?
7. What are some of the physical qualities of the cast alloys?
8. How did super-high-speed steel compare with other known cutting tools?
9. What single factor has contributed substantially to America's capacity for production?

10. What is the story behind "powder metallurgy?" What effect did it have on industry?

11. What are the steps involved in the production of tungsten carbide? What were some of the subsequent developments?

12. What is the "network phenomenon?"

13. What is a "eutectic alloy?"

14. What is negative rake and how does it affect the machining problem?

15. Describe the S.A.E. system of designating steel.

16. What have been the arguments, pro and con, regarding the use of coolants with carbide cutting tools?

17. What have been some of the problems arising out of high-speed operation?

18. Why is Eli Whitney an important figure in American industrial history?

19. What effect did woodworking techniques have on the machining of the light metals?

REFERENCES

Frommelt, H.A. "The Art of Metal Cutting," Machine Tool Blue Book, Chicago: October, 1944, to June, 1945.

MacKenzie, A. "Cemented or Sintered Hard Carbides," Metals Handbook, Cleveland: American Society for Metals, 1942 revision.

McKenna, Philip N. "Tool Materials (Cemented Carbides)," Powder Metallurgy, edited by John Wulff, Cleveland: American Society for Metals, 1942.

Engle, E. W. "Cemented Carbides," Powder Metallurgy, Cleveland: American Society for Metals.

CHAPTER II

Converting to Carbides

Requirements of Conversion. What was said in the last chapter regarding the need for redesigned machines to take full advantage of carbide tools should not be taken to mean that carbide tools cannot be used efficiently on the machines now available. As a matter of fact, a carbide tool in use on almost any machine, no matter of what vintage, will, in 99 cases out of 100, add efficiency, speed, and savings to the operation.

There are many machine tools now on the market that have been designed especially for the use of the carbides. While few of them fulfill the ideal and none of them has yet made it possible for users to find or even forecast the upper limits of productive speed, they are far ahead of anything available ten years ago and improvements are certain to be made from time to time.

However, this chapter is concerned with methods and means of using carbides on machine tools presently in use. For the most part, the needs are rather simple. There are a few points to keep in mind, nevertheless, in considering these matters. First, as has been pointed out, carbides allow machines to be run at greater speeds. Actually, the use of carbides demands greater speeds. It doesn't just allow them. Further along it will be shown how the carbides cut at various speeds and the reason for that statement will be apparent. Second, the use of carbides allows deeper cuts to be made. Again, in some operations, deeper cuts are demanded, not merely permitted. So, if machines are going to run faster and take off more material at each revolution, a third consideration enters the conversion picture--more power. And, if negative rake cutting is to be done where chip thickness is increased two or three times over what is considered "normal," there will be the need for even more power.

Finally, among the general requirements for the use of the carbides is rigidity in the machine. It must be remembered that while carbide materials are exceedingly hard, they are also quite brittle. Any chatter, wobble, or jerkiness in machine operation will quickly ruin the tools and cost more than their use has gained. Centers, spindles, and arbors must run smoothly and true, and at higher speeds than they probably were ever operated before. Hence, there is the need to look into the matter of bearings and lubrication. It may even be helpful, perhaps,

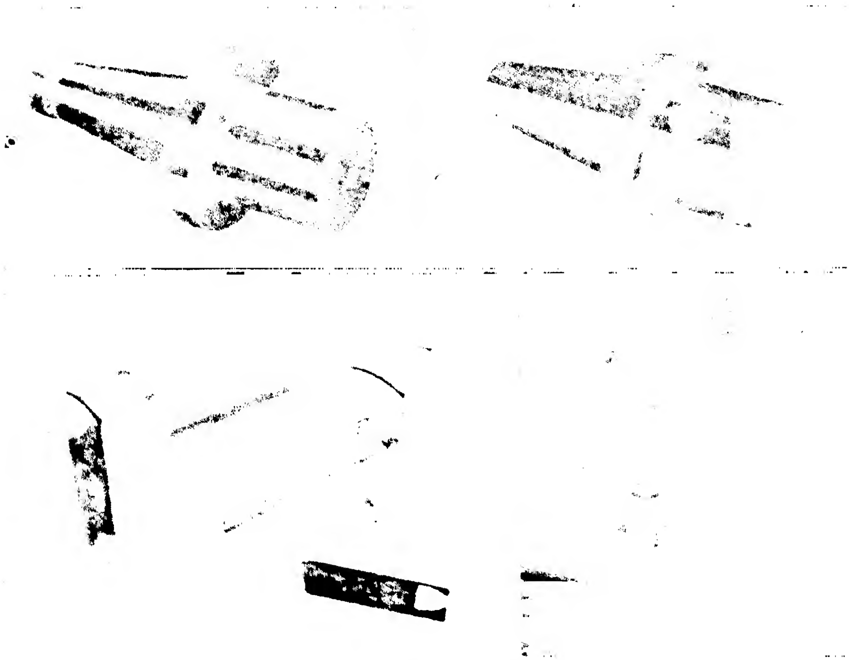


Fig. 1. Special Milling Cutters Available for Standard Machines

Upper Left, a Single-Point Fly Cutter; Upper Right, a Double-Point Fly Cutter Lower Left, Front and Rear View of a Solid Shank, Tri-Bit Fly Cutter, and Lower Right, a Shell-Type Fly Cutter with Flywheel in Place on a Milling Machine

Courtesy of Weddell Tools, Inc

to improvise some sort of satisfactory cooling system for these parts.

Carbides and Milling Machines. Part of the rigidity and smoothness of operation usually held to be essential in this equipment is achieved through the use of a flywheel. In practically all milling operations, unless the cutter itself is large and heavy, a flywheel is needed. The only exceptions occur where single-point-tool machines are used which operate at much slower speeds anyway, and where the fitting of any sort of flywheel is impossible.

In milling, practice has shown that steel, aluminum, magnesium, and almost any other metal is best machined with coarse-toothed cutters. This is because most machines either lack sufficient power or the setup will not stand the high pressures of closely spaced teeth. If tooth spacing of the usual closeness is used, the chip load is so small that the job is inefficient. Thus, for most jobs it will probably be desirable to use a cutter with one, two, three, or four blades. Special cutters have been evolved and are now available for standard machines. Various examples of these are shown in the shell and shank "fly" cutters pictured in Fig. 1.

TABLE I. CHIP LOAD PER TOOTH

Duty	Maximum Depth (in Inches)	Chip Thickness
Medium	1/4	.005 to .010
Heavy	1/2	.008 to .012
Extra heavy	3/4	.012 to .020

Information is given in Tables I and II which will allow one to figure with reasonable accuracy how to operate his present equipment with carbide cutters. It should be remembered, however, that the degree of finish wanted, the condition of the material itself, the part or design to be milled, the rigidity of the machine, and the power that is available are all matters which should be taken into consideration. They are factors which no one can determine except the person concerned.

Experience has indicated that one hp is required to remove 1/2 cubic inch of mild steel per minute, one cubic inch of cast iron per minute, 1/3 cubic inch of alloy steel per minute, and three cubic inches of aluminum per minute. Stated in another way, two hp is required to remove one cubic inch of mild steel per minute, one hp is required to remove one cubic inch of cast iron per minute, three hp is required to remove one cubic inch of alloy steel per minute, and 1/3 hp is required to remove one cubic inch of aluminum per minute.

In connection with this data and that given in Tables I and II, these operating formulas are offered:

$$\text{Cu. in per min. of metal removed} = \frac{\text{hp available}}{\begin{array}{l}\text{2 (for mild steel)}\\ \text{or 1 (for cast iron)}\\ \text{or 3 (for alloy steel)}\end{array}}$$

$$\text{or hp} \times 3 \text{ (for aluminum or copper)}$$

$$\text{Feed per minute in inches} = \frac{\text{cubic inches per minute}}{\text{depth} \times \text{width of cut}}$$

$$\text{The r.p.m. (approximately)} = \frac{\text{cutting speed (f.p.m.)} \times 4}{\text{diameter in inches of cutter}}$$

$$\text{The chip load} = \frac{\text{feed in inches per minute}}{\text{number of blades} \times \text{r.p.m.}}$$

$$\text{Number of blades in cutter} = \frac{\text{feed in inches per minute}}{\text{chip load} \times \text{r.p.m.}}$$

It should be noted here that Tables I and II are not intended to be complete. They merely represent a starting point for the operator who is faced with the problem of a quick conversion to the use of carbide

TABLE II. CUTTING SPEEDS IN SURFACE FEET PER MINUTE

Material in Cutter	Material Machined			
	Medium or Soft Cast Iron	Alloy Steel, Hard Cast Iron, Cast Steel	Mild Steel, Malleable Iron, Brass	Aluminum Magnesium
High-speed steel.........	60	45	90	750
Super high-speed steel.	75	60	xx	1000
Cast alloy....	100	90	200	1500
Sintered carbide..........	200	150 to 800	300 to 1000	1500 and up

tools. Detailed data will be presented in subsequent chapters for those who intend going fully into the various points entailed.

It is obvious that these coarse-bladed cutters will impart an intermittent action as they bump in and out of the cut. Unless something is done to dampen such blows, heavy strains will be put on the drive, the machine, the cutter, and the work. It is for this reason that flywheels should be placed somewhere in the drive. Entirely aside from its dampening effect, the energy of the spinning flywheel adds considerably to the force needed to push the blades through the cut. The purpose of any flywheel, of course, is to store up power and to distribute it more evenly.

It was stated a moment ago that the flywheel should be included "somewhere" in the drive. That was correct, as far as it went, but the closer the flywheel is to the cutter itself, the better it will do both jobs for which it is intended. If the wheel can be placed on the arbor beside the cutter, or with a flange extending partly around or over the cutter, the wheel will work better and will eliminate much of the "whip" in the drive.

As a simple means of placing flywheels in the cutter drive, flywheel arbors are offered by a number of manufacturers. In these, two types of which are shown in Fig. 2, a large flywheel replaces the usual shell end-mill arbor. It is made with a counterbored back or locating devices so it will fit on any milling machine. The face mill, shell end mill, or fly cutter is fitted or fastened to a nose, integral with the wheel, and similar to the shell end-mill arbor nose. Where cutters fit directly onto the spindle nose, a flywheel spindle-nose adapter, such as those shown in Fig. 3, may be used. The back is adapted to fit the spindle nose of the machine so that a standard cutter can be mounted.

On some machines such as the planer-type millers, the cutter is centered by a shank and driven directly by the spindle keys. This setup and the sort of flywheel which will work best with it are shown in Fig. 4. Here, a special, tapered, centering arbor with an extended nose is used,

Fig. 2. Two Types of Flywheel Arbors in Which the Flywheel Replaces the Usual Shell and Mill Arbor
Courtesy of Weddell Tools, Inc

while a flywheel is placed between the nose cutter and the face plate. This type of flywheel is made with keys and keyways to transfer the power directly to the cutter. Bolt holes are provided so that either the wheel or the cutter may be retained independently of the other.

In milling operations there are, of course, more points to watch in converting to carbides than with the machines using single-pointed tools. A knee-type milling machine of late model is shown in Fig. 5. All clamp and lock locations have been marked for convenience. A smaller machine of different make is shown in Fig. 6. Again, the

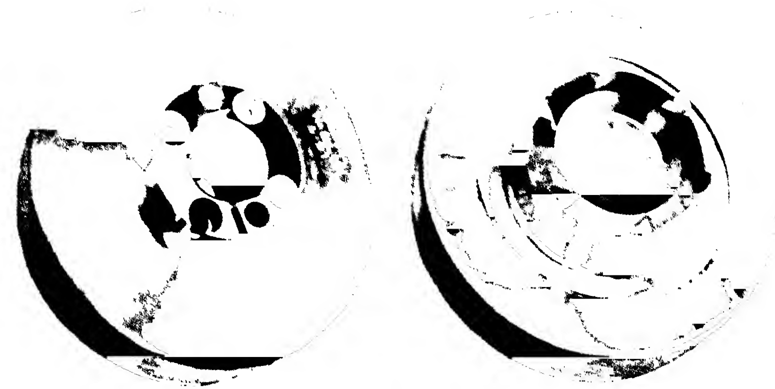
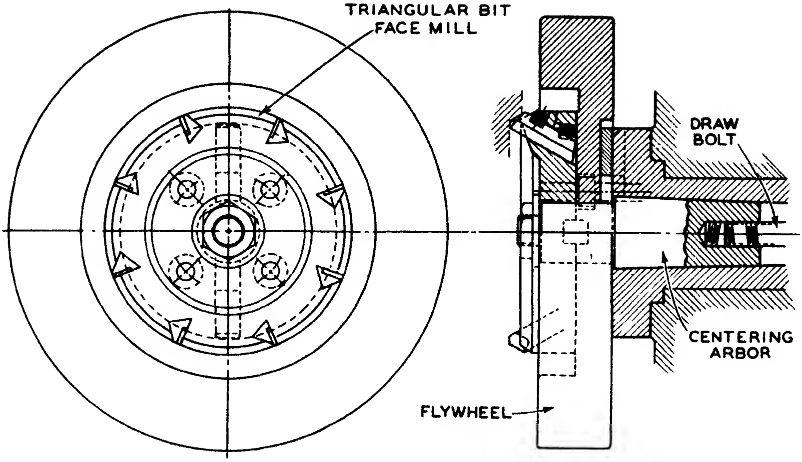


Fig. 3. Two Types of Flywheel Spindle Nose Adapters
Courtesy of Weddell Tools, Inc



FLYWHEEL SETUP FOR PLANER MILL

Fig. 4. A Flywheel on a Centering Arbor as Used on Planer-Type Milling Machines

Courtesy of Weddell Tools, Inc.

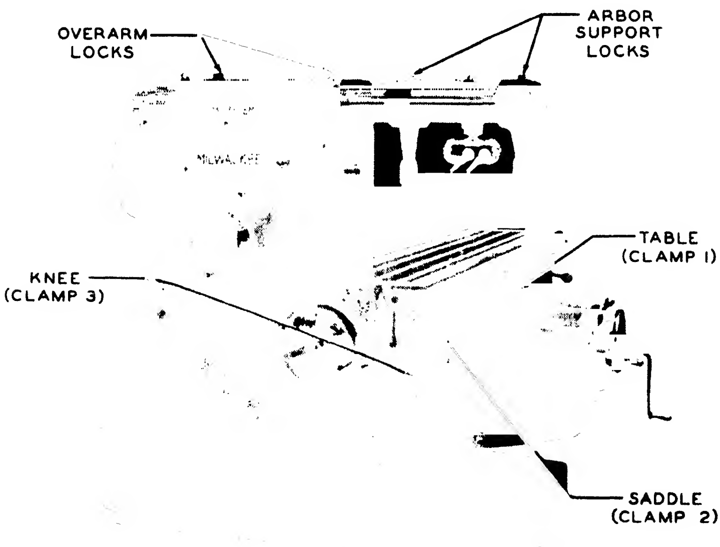


Fig. 5. An Example of One of the Knee-Type Milling Machines

Courtesy of Kearney & Trecker Corp.

important features—the clamps and gibs—are clearly indicated.

The horizontal, knee-type milling machine comes in two types: the plain and the universal. The latter type is used chiefly for toolroom work such as the cutting of spirals and gears. It may also be used for carbide milling of steel but the plain model is preferred for this purpose since it is the more rigid of the two. There are certain attachments

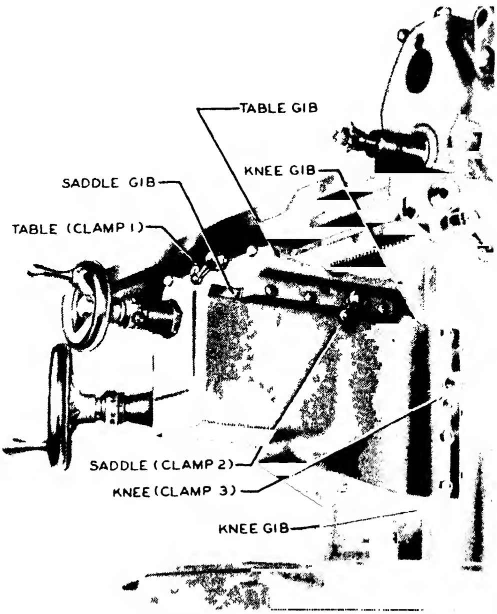


Fig 6 A Smaller Type of Modern Milling Machine
Courtesy of the Atlas Press Co

available which convert a horizontal milling machine into a vertical one which also can use the carbide technique. Again, the plain machine is preferred. When attachments are added to convert to vertical use, they should always be of the heavy-duty variety.

Perhaps the first thing to check on a machine is the proper adjustment of the gibs. In Fig. 6, the location of table, knee, and saddle gibs is shown. All of these must be adjusted properly to assure the utmost rigidity before carbide milling is begun. Loose gibs at any or all of these points will bring about chattering and vibration which will result in a great loss in tool life.

End thrust in spindle bearings, either horizontal or vertical, is probably the next point that should be checked. If there is much play at this point, carbide tips will check and crater much earlier than they should. Clamps on the knee-type milling machines are also provided for table, knee, and saddle units for the purpose of increasing rigidity.

The operator should tighten clamps 2 and 3 (shown in Figs. 5 and 6) when feeding the table back and forth, tighten clamps 1 and 3 when feeding the table in and out, and tighten clamps 1 and 2 when feeding the table up and down. A note of caution perhaps should be injected at this point. Never move units by hand or power before the clamps are

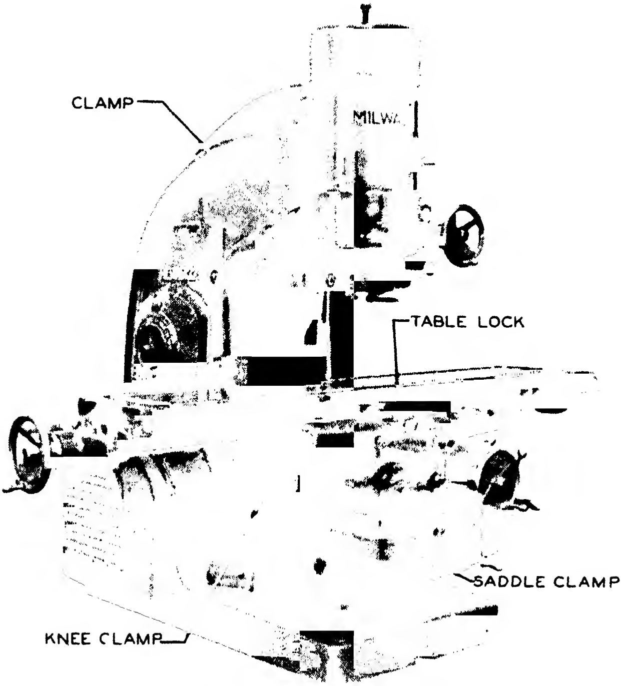


Fig. 7. Typical Vertical Milling Machine with Built-in Flywheel
Courtesy of Kearney & Trecker Corp

loosened. Moving them while the work is still clamped in the machine may result in badly scored bearing surfaces.

Slipping of any clutches must be prevented and, if necessary, a V belt drive should be provided to overcome this problem. While attending to this point it would be well to check the electrical equipment. If motor driven, the inspection should include not only the motor but the safety devices, such as relays and circuit breakers, as well.

Bushings and arbor supports (shown in Fig. 6) must be adjusted for a good running fit. If carbide tools are to be used on a vertical machine, the sliding head shown in Fig. 7 should be operated as near the upper end of its travel as possible to provide greater rigidity. At the same time, the sliding head should be properly clamped. For light and medium cuts, the head should be tightened with the clamp shown on the

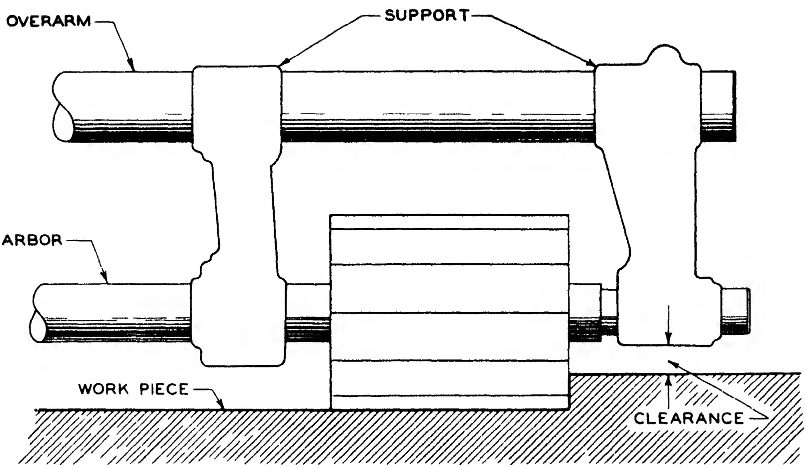


Fig 8 An Overarm Support Must Be Used When the Workpiece Size Requires That Milling Be Done Some Distance from the Machine

Fig 9 Illustrating Correct Use of Overarm Supports
Courtesy of Kearney & Trecker Corp

left side of the head in Fig. 7. For maximum rigidity during heavy, face-milling operations, all three clamps and the table lock should be tightened. If attachments are added to a milling machine, they should be applied with the overarms properly extended for complete support and absolute alignment.

Odd as it may sound in a machine shop, cleanliness in such assemblies as the arbor and spindle is one of the essentials of good milling practice with carbides. Not only should the arbor and spindle be kept clean but points between collars, bearings, and between the cutter and the arbor should be equally free from all foreign matter.

Shims for cutter spacing should be kept in perfect mechanical condition for the best results. The cutter should slide on the arbor easily, but should fit snugly. A loose cutter gives a poor finish because only few teeth will cut, and may cause chatter. A key, sufficiently long to secure a narrow cutter together with bushings on both sides, should always be used with carbide tools. Where saws or narrow cutters are used, they should be supported on both sides by large diameter collars to insure rigid operation.

If the table is kept close to the column face, rigidity and stability will be added to the machine. The type of arbor and support shown in Fig. 8 should be used when the clearance between the arbor and the workpiece or the fixture is sufficient to allow for its use. If an arbor such as shown in Fig. 9 is used for heavy cuts, it should be properly supported for the best milling results. Workpieces should always be held down securely, either by the clamps provided on the machine or by others devised by ingenuity. How this may be done on difficult jobs is shown clearly in Fig. 10. In any case, the workpiece must be supported tightly and rigidly before the cut begins. A stop bolted to the table often will help to hold the workpiece in position. Such an installation is shown in (B) of Fig. 10.

It may be necessary to place a piece of paper under a long, thin workpiece as in (C) of Fig. 10 in order to prevent the distortion which can result from the clamping. The clamp stud always should be placed close to the workpiece for effective holding. Shims should be used between the finished surfaces of the workpiece and the clamps. This is illustrated in (D) of Fig. 10. If shallow workpieces are held in a vise, they should be set on parallels and, whenever possible, the job should be gripped as centrally as possible. This will help equalize the clamping pressure.

Most tool engineers prefer to use "climb" milling with carbide tool rather than the usual "milling up." In climb milling the chip thickness is at its maximum at the beginning of the cut. Because of this, the carbide tip is subjected to less wear and heating action. This is shown graphically in (E) of Fig. 10.

Carbides and Large Machines. For a long time there was little interest among operators or tool engineers in the use of carbide tools on such large machines as boring mills; big, vertical, turret

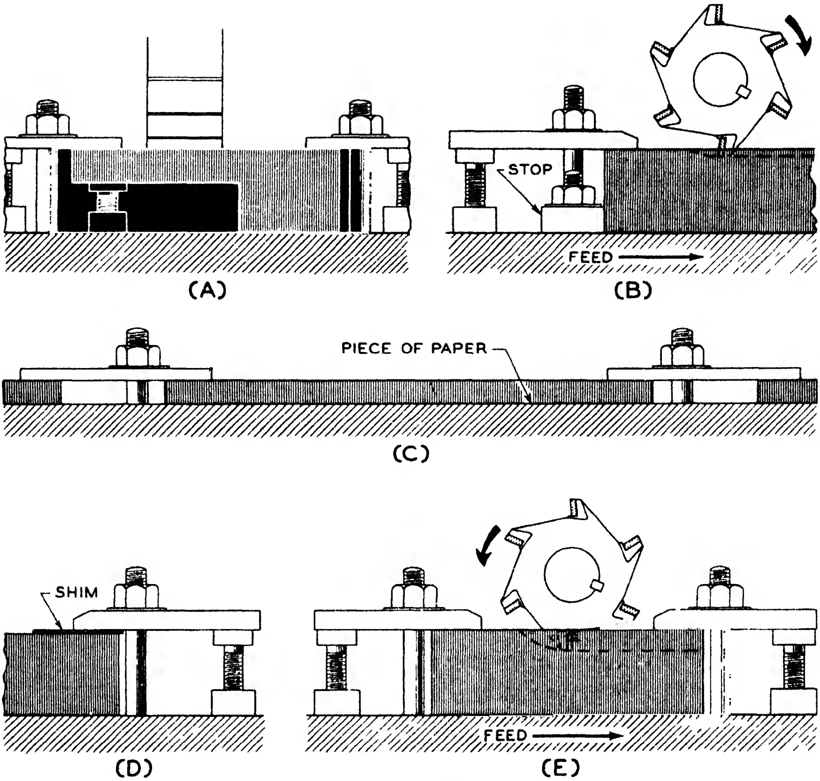


Fig. 10. Collection of Useful Setup Aids

(A) How a Difficult Clamping Job May Be Solved
(B) How a Stop Bolted to the Table May Aid in Clamping
(C) A Piece of Paper under a Long, Thin Workpiece May Prevent Distortion from Clamping
(D) Shims Should Be Used between the Finished Surface of the Workpiece and the Clamps
(E) Why Chip Thickness Is at Its Maximum at the Start of the Cut in "Climb" Milling

lathes; heavy engine lathes; and other machine-tool equipment in this class. These large machines were rarely operated on any sort of a continous basis. If they were, the machines were apt to be of an older vintage where 25 to 30 feet a minute cutting speed was about all that was expected of them. Most of them, also, were not powered heavily enough to make efficient use of the carbides and they often lacked the rigidity needed for such high-speed operation.

These conditions were changed by World War II. Many of these large machines went on the critical list. More of them had to be manufactured without much concern being given to even minor changes in

their design. However, machines manufactured in the last few years did have more rigidity, their bearings were designed for somewhat higher cutting speeds, and many of them were produced with more available power.

About all that can be done to convert a real old timer, especially a lathe, to carbides is to throw away any fancy tool holders with which it

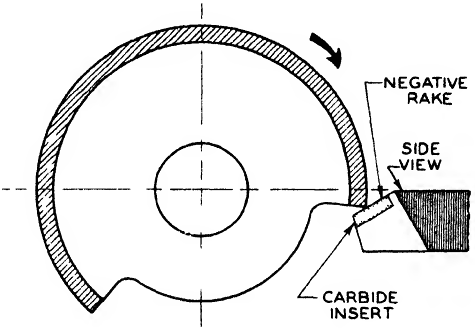


Fig. 11. The Shear Type of Cut That Results from Negative Rake Angles

may be equipped, slip a good, heavy, large-shanked, carbide-tipped bit directly into the tool post, and run the machine with as heavy a cut and as high a rate of feed as the available power and aged bearings and centers will permit. Some of the newer and greatly improved clamping

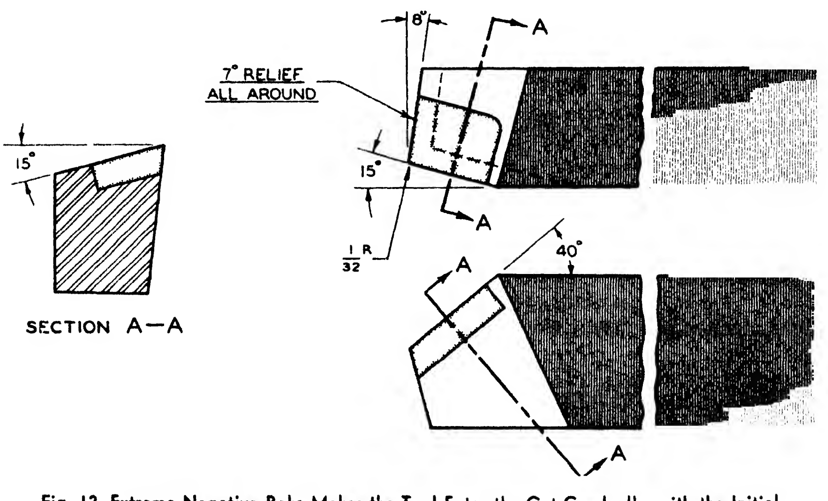


Fig. 12. Extreme Negative Rake Makes the Tool Enter the Cut Gradually, with the Initial Impact Falling Far Back from the Nose

fixtures can be used to best advantage in this conversion method.

One important consideration that will be found helpful is the proper selection of tip and shank sizes and the correct choice of tool shape and cutting angle. Negative rake will prove advantageous, even on the older machines, if the job calls for an uninterrupted cut. The "shear" type of cut which is obtained with a negative rake angle is shown in Fig. 11. It will be seen from this sketch and from that in Fig. 12 how extreme negative rake makes it possible for the tool to enter the cut gradually because the initial impact is taken far back from the nose of the tool. The point of impact would be about where line A-A in Fig. 12 crosses the edge of the carbide tip. The line of impact, of course, would depend on the angle of the tool itself. The shear cut shown in Fig. 11 is best suited to roughing cuts on pieces with large diameters. Since the chip curls toward the work and may spoil the finish, this type of cut is not recommended for finish jobs.

Another point to be considered in carbide conversion is the setting up of a centralized grinding department, assuming one has not previously been established. Such a department will result in an increased efficiency since the work done there will be performed by a specialist at grinding. The machine operator will not be expected to be as much a "jack of all trades" as formerly and will be permitted to devote more of his time to production.

It will be found, in using carbides on older machines of the single-point variety, that it is best to hold down the feed rates and let the cutting speed rise proportionally. This will make it possible to use lighter fixtures which will be simpler in design, less costly, and more easily built. Using carbides, a machine can be expected to operate two to four times as fast as when high-speed steel bits are used, providing the machine can take the punishment. Operating a machine at that speed, however, may make it necessary to hold feeds to less than .030". Although this feed rate may be smaller than ordinarily employed with high-speed steel tools, the higher footage rate will compensate for it.

Carbide bits will also allow the harder materials such as armor plate and the heat-treated steels of from 300 to 400 Brinell hardness to be machined. Previously, these metals were impossible to work. Another point worth mentioning is that operators will be able to turn out a great deal more work with much less effort. They will not have to change tools nor regrind them as often. Neither will they have to constantly check dimensions to make sure that tool wear has not changed the setup.

Other Points to Check. The next point to check in converting older and larger machines to carbide is to make sure that belts, clutches, and similar devices are able to transmit the required horsepower to the spindle. Clutch fingers should be adjusted to prevent slipping and stalling. If a machine is designed for a flat-belt drive, it may be desirable to change to a V belt system. When this is done, the number of belts should be great enough to transmit the load.

CONVERTING TO CARBIDES

It will be recalled that mention was made in Chapter I regard the difficulty with centers on lathes fitted with carbide tools. Some of antifriction tailstock center should certainly be used on any lat fitted. The type and make is a matter of individual judgment.

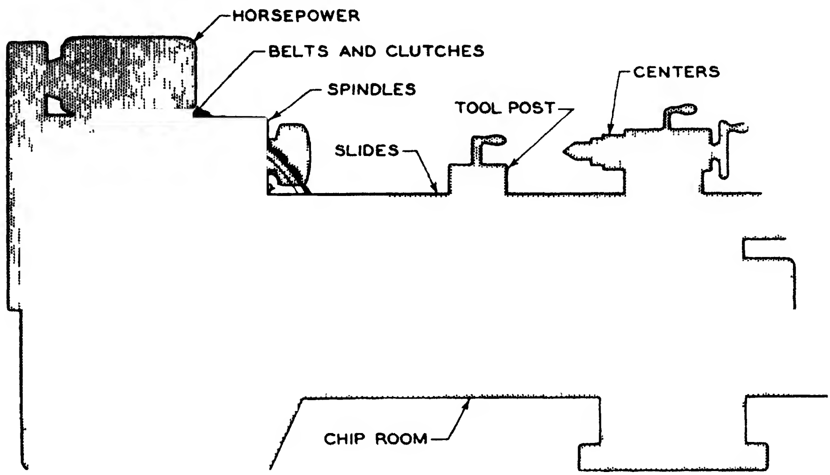


Fig 13 Points to Check When Converting Larger Machines to Carbides

If the openings in the machine bed and around the tool holders or blocks are too small to allow the greatly increased flow of chips to get away without difficulty, a chute built of sheet metal may prove to be the answer. Chip breakers are sometimes added if the size of the openings demands the production of smaller chips.

Bearings, slides, and ways should be checked for wear to prevent backlash and chatter. Most of the check points mentioned are shown in Fig. 13. A trick that has proved helpful to quite a few operators and tool engineers who found it impossible to do anything about the worn condition of their machines, is that child of dispute, negative rake. It will be found that negative rake will end a lot of chatter when there is nothing else that can be done about it. Of course, negative rake introduces its own problems which will be discussed further on.

Carbides and Shop Efficiency. Many antiquated practices that have grown up with a shop will have to be cast aside when carbides are put to work. The engineer in the larger shop or the owner of a smaller establishment may derive much benefit from such a move. Among the new or different methods which may be found helpful as a result of the introduction of carbides into a shop, are real planning of the production cycle, and the adoption of some kind of tool-setting equipment which would decrease setup time.

Planning the production cycle may mean no more than changing the

ıer of the cuts so that the unwanted metal may be removed in the ·est and most efficient way. Many shops, particularly the small ones, ·r have given any real time or mental effort to this matter. An ex-

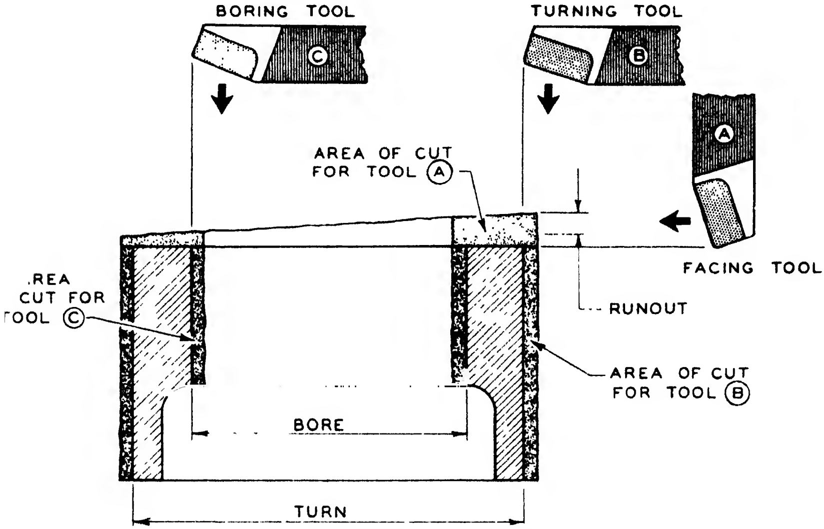


Fig. 14. How Good Planning Will Aid Production

cellent example of the way real planning works may be seen in Fig. 14, where ʼool A takes a facing cut to remove scale and runout on the back of the casting. Tools B and C then take the light, preliminary turning and boring cuts, enabling the operator to hold dimensions to closer tolerances throughout the whole length of the cut since the tools start in clean metal. All this reduces excessive runout in the work. It may reduce not only the travel time of both the turning and boring machines but also the handling time since there is less time lost in changing tools. Cutting a 30-hour machining job to one of eight or ten hours through intelligent planning and the use of carbides was not at all unusual as a result of wartime necessity.

The use of tool-setting equipment to speed setup procedures is a point that can add greatly to production. Some authorities have urged the adoption of a "get rid of calipers" campaign, with subsequent faster setup through the use of dial indicators, machine dials, and simple setting gages as part of the fixture itself. If the operator will check the machine dial setting when changing tools, he will find that it will speed setup greatly. Unfortunately, many operators have no faith in these dials. Previous unsatisfactory experience with older and less accurate machines has conditioned them against such "gadgets." They fail to recognize that on the reasonably late machines, those with ground lead screws and dials that are accurately marked, the situation has changed.

A good machine operator should keep pace with the advancements that are made in his field.

All in all, if a machine can run fast enough and smoothly, it can use carbide tools, regardless of its age. Cutting speed must be high enough to prevent excessive cratering as well as to check the formation of a built-up edge on the tool if the full advantage of the use of the carbides is to be realized. Since all of these matters will be discussed in detail in subsequent chapters of the book, there is no point in going further into their study here, except, perhaps, to explain that cratering is the hollowing out of the carbide tip at a point behind the cutting edge where the chip strikes the tip and tends to pull off pieces of the carbide.

Power Formula for Carbide Conversion. The problem of adequate horsepower may require some study. For this reason, a simple rule is offered at this point which can be used to check the horsepower needs for any given job. A few typical examples of horsepower requirements were given in this chapter in the discussion of the carbides and milling machines. However, the information was general. Using this simple power formula, the requirement for almost any given job on steel can be determined with fair accuracy: horsepower per tool required equals the depth of the cut in inches times the feed in inches times the surface feet per minute times a power constant which is selected from Table III. According to Table III, the power constant will vary between

TABLE III. POWER CONSTANTS FOR STEELS

S.A.E. Designation	Constant
1010-1025	6
1030-1095	8
1112-1120	6
X1314-X1340	8
T1330-T1350	9
2015-2320	7
2330-2350	9
3115-3130	8
3135-3450	9
4130-4820	9
5120-52100	10
6115-6195	10
Cast steel	9

six and ten, depending upon the steel to be cut. The power to run the machine with the tools not cutting must be added to the figure obtained to get the total horsepower requirement. The horsepower required merely to turn the machine, without cutting, usually is figured at about 30 per cent of that required to make the cut. Stated as a formula, the rule becomes:

$$hp = d \times f \times S \times C$$

where d is the depth of the cut in inches, f is the feed in inches, S is the cutting speed in feet per minute, and C is the constant. Stated algebraically, the formula would be:

$$hp = dfSC$$

It is obvious that any one of the four figures on the right side of the equation can be solved for if the other three are given. For example, suppose that the available horsepower, the depth, the cutting speed, and the constant are known and it is desired to find the proper feed. By simple algebraic transposition, the formula becomes:

$$f = \frac{hp}{dSC}$$

The reason for all formulas is to facilitate the process of finding a missing figure by simple algebra or arithmetic. Formulas will be given in this book wherever there is a need for them. They may grow in complexity, but the basic process will remain the same.

Applications of the Carbides. Chief reasons for America's greater ability to produce machinery, appliances, and other commodities, are the use of high-speed, automatic machines; modern plants designed and organized for a steady flow of production; scientific use of labor permitting one operator to supervise many machines; the measuring of machines in terms of output; and, finally, the proper selection and use of tools at speeds heretofore unknown. The tools that will perform best at high speeds are those tipped with sintered carbides.

In the mass production of such parts as gears, locomotive drive wheels, pistons, machine spindles, cylinders for steam and internal combustion engines, steam turbine shafts, steam and gas engine crankshafts, camshafts, studs, machine frames, and thousands of other articles, the sintered carbide tools can increase production from 50 to 200 per cent. This increase may even be higher when the volume of work produced is very large, since with carbide tools, the cutting speed is from three to six times that of high-speed steel. At the same time, the added tool life between grinds and resetting results in increases of from three to ten times the production obtained using high-speed-steel tools.

Carbide tools are essential for cutting soft materials, particularly in the so-called mass production industries. The high speeds at which they operate and their long tool life makes them particularly valuable where production and assembly lines move rapidly.

Sintered carbide tools are of great value in cutting tough, strong, hard metals such as nickel, chromium, molybdenum, vanadium and manganese steels, stainless steels, and monel metal. They are particularly adaptable to the cutting of all alloy steels which have been heat-treated to a hardness of from 200 to 500 Brinell or higher, and they make possible the finishing of such surfaces by turning instead of grinding. It is true that heat-treated pieces of greater hardness may sometimes be machined with high-speed tools, but at speeds from 15 to 20

f.p.m., as against 75 to 100 f.p.m. or more, for carbide tools. Turning heat-treated steel to the required dimensions instead of roughing it out with turning tools and grinding it to the finish, results in great savings. The carbides remove unwanted material rapidly, yet leave a finish that compares favorably with grinding under the circumstances.

For machining castings made of steel, gray iron, malleable iron, brass, bronze, and phosphor bronze, carbides are superior since they can withstand the abrasive action of scale usually present on castings and the sand inclusions often found in them, much better than can the high-speed-steel tools. Here again, the cutting speed is often from four to six times the rate possible with high-speed steel.

Chilled-cast-iron castings can be machined with comparative ease by using carbide tools. The carbides used for this purpose are the straight tungsten carbides, milled and sintered with cobalt as a binder.

Outstanding performances are obtained with carbide tools in facing, boring, and turning steels that have been heat-treated up to a hardness of 600 on the Brinell scale. Spindles, studs, gears, and similar parts may be heat-treated and machined in the hard state instead of being roughed out in the annealed state, heat-treated, and then ground to the required size. Hard and tough alloys such as the austenitic stainless steels, Hatfield's manganese steel, Stellite, and Hastelloy may now be machined with little difficulty when proper precautions are taken as to cutting speeds, feeds, depths of cuts, and the cutting angles on the tools.

Carbide Tools in Jobbing Shops. In jobbing shops where manufacturing is done on a small scale, the use of sintered carbides is just as desirable as it was in the larger shops because, as was shown in connection with the heat-treated steels, dimensional accuracy is more easily attained, and a higher surface finish is possible without any grinding being necessary. The use of carbide tools in a jobbing shop will prove to be economical and the savings resulting from this use may be considerable. This is especially true when one considers that frequently, when working on general-purpose machines, only one tool is used on the machine at any one time. Thus, the setup time is shorter than when production machines with multiple tool setups are used.

In repair work, parts are frequently found which have either been heat-treated or work hardened to such an extent that high-speed tools will not perform satisfactorily. In such cases, the carbides afford a means of machining these parts. Again, when weldings must be machined, carbide tools with back and side rake will do the work satisfactorily.

Carbide Tools in Mass Production. Mass production requires the use of single-purpose machines. These are frequently designed for carbide cutting tools. Once the setup has been made, tools last a long time and retain keen cutting edges. They permit work to be produced at high surface rates and to required dimensions without frequent interruptions for tool changes. Where such conditions prevail, tools made of other materials cannot compete successfully with carbides.

Fig. 15. Setup for Machining a Cast-Iron Pulley
Courtesy of the Carboloy Company, Inc

Examples of Efficient Use of Carbides. Here are several typical machining setups which not only show how and where carbides should be used but demonstrate their effectiveness as well. The first example is the setup on a turret lathe for roughing and finishing the groove on a cast-iron pulley, facing the ends, and spotfacing the hub, all of which is done in one operation. This job is shown in Fig. 15. The pertinent data for this operation is as follows:

Performance		Pieces per Grind of Tool	
r.p.m.	245	Groove	2,450
Depth of cut	1/8"	Face	5,000
Feed per rev.	.026"	Spotface	35,000
		Machine output in eight hours	1,000

Examples of the use of carbides in machining armor plate, S.A.E. 1020 forged and normalized steel, steel gear blank forgings, and hardened ball-bearing races of heat-treated steel of 650 Brinell hardness, are given in Table IV.

These examples are by no means complete as to the work being done by the carbide-tipped tools today, nor are they necessarily representative. They are merely a partial cross section of the developments in the metal working field which have come about in the last few years and which have written new chapters in the history of the industry.

TABLE IV. SPECIFIC MACHINING DATA INVOLVING CARBIDE TOOLS

Material to Be Machined	Cutting Speed in f.p.m.	Depth of Cut (In.)	Feed per Rev. (In.)	Back Rake of Tool	Side Rake of Tool	Radius of Tool (In.)
Turning armor plate	250*	1/8	.025	45° (neg.)	5° (pos.)	1/16
	250-300†	1/16	.015	10° (neg.)		1/16
S.A.E. steel	500	1/4	1/64	10° (neg.)	5° (pos.)	1/16
Steel gear forgings	300	1/4	.025	10° (neg.)	10° (neg.)	1/16
Hardened bearing races	200-250	—	—	10° (neg.)	—	3/16

*Roughing cut.
†Finishing cut.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What are the four principal requirements in putting carbide tools to work?
2. Where would special cooling systems be applied to machines using carbides?
3. What is the purpose of a flywheel?
4. Where would a flywheel be installed on a lathe equipped to use carbide tools?
5. Why should coarse-toothed cutters be used in milling with carbides?
6. What would be a conservative chip thickness per tooth in heavy-duty milling at a depth of 3/4"?
7. How is the "chip load" found?
8. Where should a flywheel be placed in the drive?
9. Can carbide-tipped cutters be used in a horizontal milling machine which has been converted to vertical use by attachments?
10. What should be checked first in converting a knee-type milling machine to the use of carbides?

11. What is the second point to be checked?

12. Where should the sliding head of a vertical machine be set when using carbide cutters?

13. Is cleanliness of the machine essential when using carbides?

14. What results from a loose cutter?

15. How should a carbide saw be supported?

16. How should a long, thin piece of work be supported to prevent distortion in it as a result of clamping?

17. What is "climb" milling?

18. Where is the chip the thickest in climb milling?

19. Is negative rake of any advantage on old, single-point machines such as lathes or planers?

REFERENCES

Frommelt, H. A. "The Art of Metal Cutting," Machine Tool Blue Book, Chicago: October, 1944, to June, 1945.

Longwell, James R. "Older Equipment No Bar to Carbides for Steel Cutting," Modern Machine Shop, November, 1940.

Lucht, Fred W. "Using Carbides on Large Machines," Railway Mechanical Engineer, July, 1942.

War Production Board, Office of Production Research and Development "Data Sheets M-40, M-41, M-42," July 10, 1944.

CHAPTER III

Tool Angles, Machinability

Importance of Tool Angles and Nomenclature. There has been some confusion among operators and even among tool engineers and tool buyers as to the correct names of the various angles and clearances designed, built, and ground into carbide tools. Because of this question of correct terminology, a standard nomenclature of these tool features was compiled by the American Standards Association and is included in this chapter. In addition, the purpose of these features will be carefully described and illustrated, thus helping to clear up the disorganized conceptions existing in many shops all the way from the drafting room to the production line. How the carbide tool cuts and why it has the ability to do a better job than any of the cutting tools previously developed will also be discussed in this chapter.

This matter of tool angles is highly important, as will be evident further on. It is suggested that the information which will be presented concerning single-pointed tools be thoroughly mastered. A complete understanding of single-point tool angles will greatly facilitate the comprehension of these angles when they are applied to multiple-pointed tools and tools which turn with or against the feed of the work. Since some of these angles become quite complex, it is essential that an adequate knowledge of them be acquired.

Machines such as engine lathes, turret lathes, hand screw machines, automatic screw machines, semiautomatic lathes, boring mills, shapers, and planers are machine tools which use single-pointed cutting tools. All tools used in these machines are similar in shape except where special forms are needed for special jobs. In turning, facing, and boring operations, most of the cuts are continuous. That is, the cutting force is exerted continually on the tool point or somewhere along the cutting edge. On the other hand, in planing and shaping operations, the cutting action is intermittent. For this reason, the angles to which the tools are ground and shaped may be different from those used for turning. Again, for use in intermittent cutting, the tools should possess more rigidity than for continuous turning operations, while the rake angles may be ground with a still different inclination.

Single-Point Tool Nomenclature. A typical, single-point tool for turning operations is shown in (A) of Fig. 1. It consists of a rectangular shank, X, to which a suitable, sintered carbide blank, Y, is

brazed. The names of the various parts and angles of the single-point tool, with numbers corresponding to those in Fig. 1, are back rake angle, 1; front relief or clearance angle, 2; side relief or clearance angle, 3; side rake angle, 4; side cutting-edge angle, 5; end cutting-edge

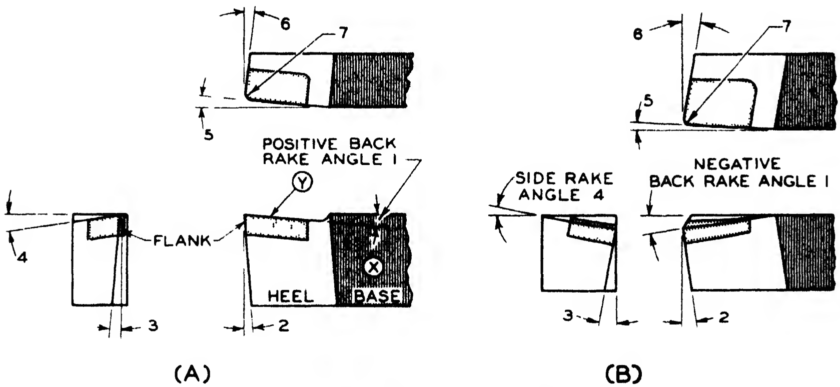


Fig. 1. Nomenclature of the Single-Point Tool

angle, 6; and nose radius, 7. Other angles are shown in (B) of Fig. 2, and in (B) of Fig. 3. These are the entering angle, 8 (Fig. 3); tool lip angle, 9 (Figs. 2 and 4); and the cutting angle, 10 (Figs. 2, 4, and 5.)

The back rake angle is shown in (A) of Fig. 2 and is designated by

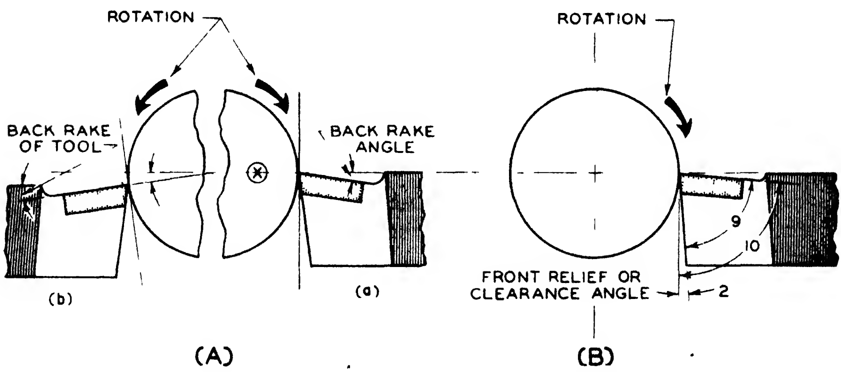


Fig. 2. Nomenclature of the Single-Point Tool

the number 1 in Fig. 1. It is that angle which measures the downward slope of the top surface of the tool from the point, or nose, to the rear. When the slope of the tool is downward from the back of the cutting edge to the front, as in (B) of Fig. 1, the rake is negative. If the slope is downward from the cutting edge of the tool to the back, the rake is positive. In both instances, the angle is measured between the face of the

tool and the perpendicular to the work as shown in (a) of (A) in Fig. 2.

The front relief or clearance angle is shown in Figs. 1, 2, and 4 and is designated by the number 2. The function of this angle is to provide clearance for that part of the tool not actually doing the cutting. It allows the tool to cut without rubbing too heavily against the work.

The side relief or clearance angle is shown in Figs. 1 and 3 and is

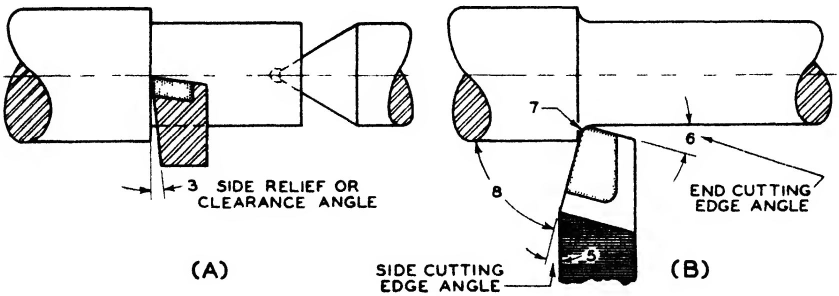


Fig. 3. Side and End Cutting-Edge Angles, and Entering Angle

designated by the number 3. The function of this angle is to provide clearance which enables the tool to advance freely into the work as shown in (A) of Fig. 3.

The side rake angle is the measure of the slope of the top surface of the tool to the side. The angle is measured in a plane parallel to the

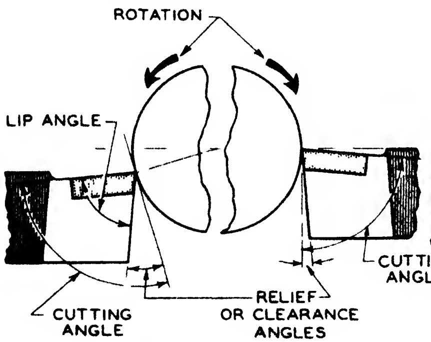


Fig. 4. How the Cutting Angle May Vary with the Setting of the Tool

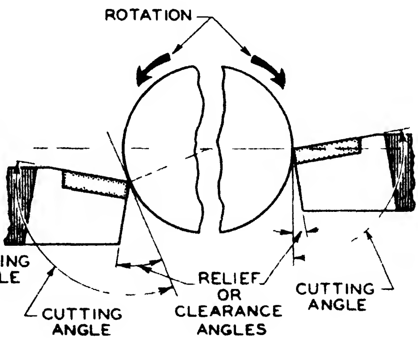


Fig. 5. Negative Rake Angle Makes Possible Heavy, Interrupted Cuts

surface of the work being cut. It may be positive or negative, depending upon the conditions of its application. This matter of positive or negative rake will be discussed more fully under the sections on the design of cutting tools. The side rake angle is shown in (A) and (B) of Fig. 1 and is designated by the number 4.

The side cutting-edge angle of the tool is the angle the cutting edge

makes with the side edge of the shank. The application of the side cutting-edge angle is shown in (B) of Fig. 3. The angle is designated by the number 5. Its function is to distribute the cut over a greater length of cutting edge, thus providing a better distribution of the cutting force exerted.

The end cutting-edge angle is the angle between the cutting edge on the end of the tool and the surface being cut. At (B) in Fig. 3 is shown how the angle is formed. Its function is to allow the tool to clear the work, preventing it from rubbing on the newly cut surface. This angle is designated by the number 6.

The nose radius of the tool is the curve formed by joining the side- and end-cutting edges. The nose radius has a definite relation to the performance of the tool insofar as finish of the work and tool life are concerned. It is designated by the number 7, and is illustrated in Figs. 1 and 3.

The entering angle is designated by the number 8 in (B) of Fig. 3 and varies with the setting of the tool shank or with the design of the tool. It will be discussed in detail later on.

The tool lip angle, designated by the number 9, is illustrated in (B) of Fig. 2 and in Fig. 4. It is the angle of the point of the tool as measured between the face and the flank in a plane at right angles to the cutting edge in contact with the work. The strength of the tool, its ability to conduct the heat away, its cutting performance, and various other matters are dependent upon this angle.

The cutting angle, designated by the number 10, is equal to the lip angle plus the relief or clearance angle as shown in Fig. 4. It will vary with the setting of the tool, depending on whether it is at the center, below center, or above center. With a negative, back-rake tool, the cutting angle may be greater than 90°, making the tool more capable of taking heavy and interrupted cuts. This is shown clearly in Fig. 5.

The flank of the tool is the ground surface below the cutting edge and is shown in (A) of Fig. 1. The base is that side of the tool which bears against the support and takes up the pressure of the cut. The heel is that part of the tool shank which is directly below the cutting edge and is also shown in (A) of Fig. 3.

In addition, there is the matter of the true rake angle which is not shown on any of these drawings. The true rake angle is the actual slope of the tool face from the cutting point or edge, and may be measured by noting the direction of the chip flow. True rake is the combination of back- and side-rake angles, together with the depth of the cut and the feed of the tool in the direction of the cut. Provided the other values remain constant, the angle varies only with a change of feed and depth of cut. The direction of chip flow also will change with variations in feed and speed.

These angles vary, of course, with the conditions under which the tool will be used. There are many factors which influence the angle to which carbide tools are ground. Some of these are the material

TABLE I. TOOL ANGLES RECOMMENDED FOR MACHINING VARIOUS MATERIALS

Material To Be Cut	Back Rake (Degrees)	Side Rake (Degrees)	Front Relief (Degrees)	Side Relief (Degrees)
Aluminum alloys.......	10-20	10-20	8-12	6-10
Aluminum castings....	10-20	10-20	6	6
Brass, yellow..........	0-5	2-6	4-8	4-8
Bronze and its alloys.	0-5	0-10	4-6	4-6
Cast iron:				
soft.....................	0-4	4-6	4-6	3-6
hard....................	0	3-6	3-5	3-4
chilled.................	0	3-6	3-5	3-5
malleable.............	0-4	4-8	3-5	3-5
Copper..................	0-10	8-15	6-8	6-8
Fiber.....................	0-10	6-12	6-10	6-10
Plastics	0-10	6-12	6-10	6-10
Rubber:				
hard.....................	5-15	8-15	6-10	6-10
soft......................	10-20	8-20	8-15	8-15
Steel:				
up to 200 Brinell	0-10	4-15	6-8	6-8
200-275 Brinell......	0-8	4-10	5-8	5-8
275-350 Brinell......	0-5	4-8	5-8	5-8
350-425 Brinell......	0-3	0-6	4-6	4-6
425- up Brinell	0	0	4-6	4-6
12% Mn, 1.2% C.....	0	0-4	3-4	3-4
18% Cr, 8% Ni	4-6	8-16	4-6	4-6
Zinc base die castings	0-10	8-12	6-10	6-10

that is to be machined, its properties including toughness and hardness, the machine used, the power available, the rigidity of the machine, the rigidity of the tool itself and of the tool-holding fixture, and the rigidity of the work and the work-holding fixtures.

The values for the angles given in Table I are based on general practice. They should be accepted as a guide rather than as a solution to the problems with which the mechanic and the engineer are confronted in their daily work. These values were obtained through a careful collection of tool performance data and later, through painstaking research. They were found to be the best for general application. However, when one meets with problems of a more particular nature, other angles, tried at random, may prove to be more satisfactory than the ones given in Table I. In some cases it may even be desirable to go to negative angles in order to meet peculiar machining problems.

Regardless of the material of which the cutting tool is made—tool steel, high-speed steel, cast alloy, sintered carbide, diamond—the cut-

ting action is the same. The big difference is in the speed of the cutting. The carbon-steel tool cuts at low speed, the high-speed steel tool at roughly twice the speed of carbon steel, the cast alloys at twice that of high-speed steel, and the sintered carbides at twice that of the cast alloys, or even higher.

Chip Formation in Ductile Metals. So-called chips of ductile metals are removed by shear and tear, and sometimes by flow. The shearing action is shown in Fig. 6. The material is compressed by the tool from thickness t_1 to t_2, as illustrated, and is sheared off in the form of a "chip." As indicated in Fig. 6 by line 0-0, the shearing of the chip is at approximately right angles to the tool face. In cutting ductile material, the chip is removed in the form of a continuous ribbon, either straight or curled.

The Built-Up Edge. When soft, ductile metal is cut, a small quantity of material collects or is built-up on the face of the cutting edge, adhering to it strongly. This is shown at (A) in Fig. 7, and is known as the built-up edge. How this is accumulated is explained as follows: When cutting starts, the ductile metal bearing on the point of the tool is compressed against the face by a force which, at the increased temperatures resulting from the cutting action, causes the material to stick there. In this condition, the cutting edge does no cutting itself but actually supports the built-up edge which does the cutting. This edge is an actual part of the tool during the cutting. It is the cause of rough-finished surfaces, since fragments of the built-up edge escape with the work under the tool point, marring the surface and leaving it rough.

The built-up edge varies in size and shape according to the material cut, the tool shape, the size of the cut, and the cutting speed. Because

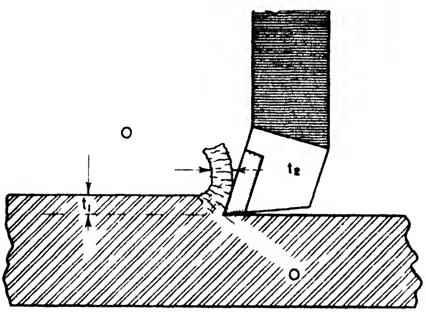


Fig. 6. Illustrating the Shearing Action of a Shaping Tool

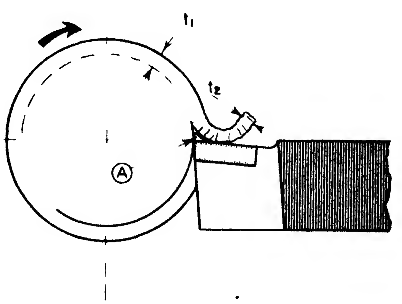
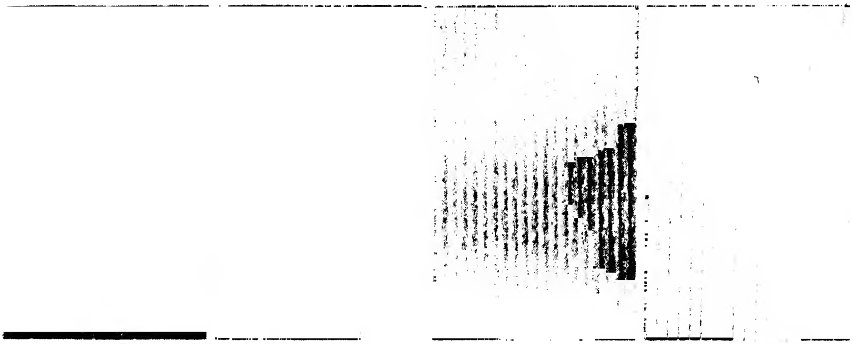


Fig. 7. Illustrating the Formation of the Built-up Edge

the built-up edge is greatest at slow speeds, the machined surface will be rough and the material removed will appear to have been torn off. At higher speeds the built-up edge is smaller and farther back from the cutting edge. As a result, the work will have a smoothly finished surface. A chip of this nature is known as a continuous chip with a built-up edge.



(A) (B) (C) (D)

Fig. 8. Speed Is an Important Factor in Producing the Desired Finish
Courtesy of Vascoloy-Ramet Corp

The peculiarities of ductile metals resulting in the machining problem just described demand that the best cutting speed be found for each metal to be machined. If the finished surface of a piece of work is rough, it will often be found that a smoother finish can be obtained by increasing the cutting speed. Experimental cutting similar to that shown in Fig. 8 will enable the most advantageous cutting speed to be found. In this particular trial, S.A.E. 3150 steel, which had been heat-treated to a Brinell hardness of 200, was cut with a Vascoloy-Ramet, steel-turning, single-pointed tool. The speed was varied from 42 to 684 f.p.m. A magnified section of the first few cuts is shown at (A) in Fig. 8. It will be noted that the surface has a torn appearance which is an indication that the speed was too low. The surface produced at 144 f.p.m., shown in (B) of Fig. 8, reveals that the torn effect has almost disappeared and that the surface of the work appears to be of good finish. The surface produced at 235 f.p.m., seen in (C) of Fig. 8, appears clean and smooth. There is no evidence of the tearing found at the lower speeds. In (D) of Fig. 8 is shown the glazed surface condition which resulted from a cutting speed which was too high. This top speed of 684 f.p.m. has produced an undesirable surface caused by the wearing off of the cutting edge of the tool. This leaves a land below the cutting edge which burnishes the surface of the work.

Since the surface produced at 235 f.p.m. resulted in the most desirable finish, this speed represents the optimum conditions under which the cutting of this particular material should be done. As might be expected, the optimum speed varies for different materials and is the factor of machinability of the material cut. It should be remembered that when cutting soft, ductile metals at high speeds, the built-up edge on the tool gets smaller as the speed increases and eventually almost disappears, giving a smooth cut with a continuous chip.

When harder ductile metals such as heat-treated medium- and high-

carbon steel or alloy steels are cut, the built-up edge is much smaller, resulting in a smooth surface and a continuous chip. This condition represents the ideal in machining operations. To obtain such a condition using carbide tools, the cutting speed must be adjusted. Excessive build-up, the result of too low a speed, should be avoided since it not only results in a roughly finished surface, but in excessive wear of the tool flank as well.

The high-speed, smooth-flowing chip is known as a continuous chip without a built-up edge. It represents the optimum in operating condi-

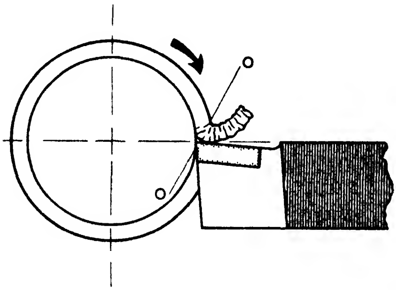


Fig. 9. The Ideal Chip

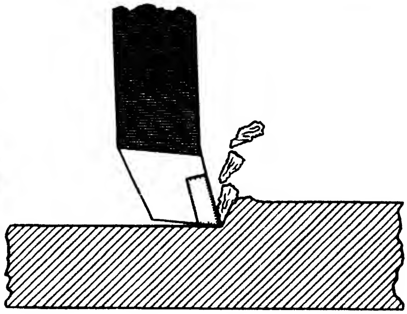


Fig. 10. Discontinuous Chip

tions and may be obtained even with relatively soft material providing the speed is sufficiently high and a cutting fluid used. The ideal chip is shown in Fig. 9.

Chip Formation in Brittle Metals. When brittle metals such as cast iron, brass and bronze, or hardened steel are cut, the chips removed are but little distorted by compression before they break from the parent metal. These chips are in small bits and are called "discontinuous." They leave the parent metal with a rough machined surface which is scraped smooth by the tool point as it removes the irregularities. Tool failure in cutting these materials is caused by abrasion between the flank of the tool and the work. The discontinuous chip is illustrated in Fig. 10.

Summing up the foregoing explanations, it may be said that there are three types of chips made by cutting tools. These are:

1. The continuous chip with a built-up edge resulting from cutting soft, ductile metals such as soft steel and aluminum with tools made of any material. A continuous chip will invariably produce a rough finished surface unless the speed is adequate and a proper lubricant used to cool the tool and work, and at the same time reduce the friction between the chip and the tool surface.

2. The continuous chip without a built-up edge resulting from the cutting of ductile metals of greater hardness. The action gives a smooth, finished surface, usually attributed to the absence of an excessive built-up edge. The use of a lubricant will aid the flow of the chip on

the tool surface, and, almost always, will prolong the life of the tool.

3. The discontinuous chip which is obtained when hard and brittle metals are cut. The action of these materials on the tool is abrasive and the tool eventually fails because of wear on the flank.

Tool Failures. Sintered carbide tools, even when properly made, maintained, and used, may fail in any of the following ways:

1. Through the development of a crater just to the rear and to one side of the cutting edge as a result of abrasion by the chip. This crater is shown in Fig. 11. As cutting action continues, the crater increases in width and depth. When its edge approaches the cutting edge of the

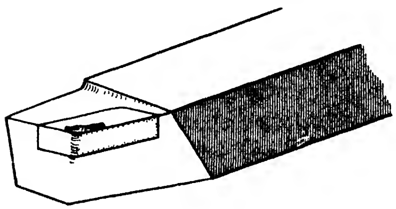


Fig. 11. Cratering of the Tip May Cause the Tool to Fail

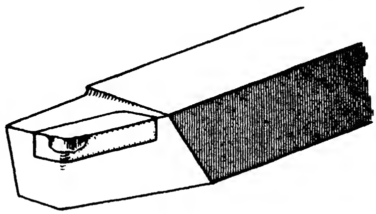


Fig. 12. How the Tool May Fail from Wear on the Flanks

tool, the tool weakens and the edge breaks out, bringing about sudden tool failure. This cratering is most noticeable when turning tough, strong, alloy steel, stainless steel, and monel metal. To obtain best results on these metals, the carbide grade must be carefully selected for each material being machined, and should be of materials possessing high cratering resistance.

2. In addition to cratering, there may be abrasion on the flanks, and tool failure may result through the combination of abrasion and cratering. This is true in cutting strong, tough, alloy steels and monel metal.

3. Through crumbling of the edges and abrasion when cutting extremely hard, strong metals.

4. Through fracture due to excessive loads.

5. Through thermal cracks, caused chiefly by improper application of cutting fluid.

6. Through excessive wear on the flanks as shown in Fig. 12. This is caused by the abrasive action of the material on the cutting edges. This condition may be brought about by too high a speed, too low a hardness of the tool material, or a combination of the two.

Effects of Tool Angles. The rakes, reliefs or clearances, and side and end cutting-edge angles all have an important effect on tool performance when other conditions remain the same. For this reason, good engineering practice in the machine shop requires that tools be carefully designed or selected for the particular job they are to be used on. Only then can the best performance be expected from the tool. The various angles found on cutting tools have been described in

the first portion of the chapter. What follows is an explanation of the purpose or function of these angles.

Back Rake. The back rake is important in that it reduces the power required to make the cut. As a result, the cutting force on the tool is reduced, extending its life. The back rake in the turning tool gives direction to the chip flow, but at the same time it reduces the lip angle of the tool, thus weakening it. This is illustrated in (A) and (B) of Fig. 13.

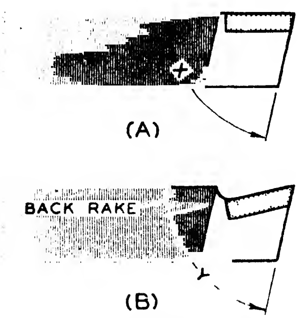


Fig. 13. Zero Back Rake (A) and Positive Back Rake (B)

A cemented carbide tool with zero back rake is shown in (A) of Fig. 13. The lip angle is designated by the letter X. A tool with a positive back rake is shown in (B), the lip angle of which is designated by the letter Y. It will be noted that the latter tool has the smaller lip angle. For turning rough forgings and steel castings and for interrupted cuts, the back rake of sintered carbide tools is usually ground to a negative angle of from five to ten degrees, as shown in Fig. 14. For shaper and planer work, however, the back rake of carbide tools may be ground negative from 15° to as much as 45°.

Negative back rake angles call for more power in cutting the metal and should not be used indiscriminately. Most manufacturers of carbide tools recommend a zero back rake for average application on common cuts. In Table I the approximate angles for turning various materials are given. An examination of this table will show that the harder materials either require less rake or that the tools be made more blunt. The purpose of this is to provide a greater lip angle and, consequently, greater strength for the tool.

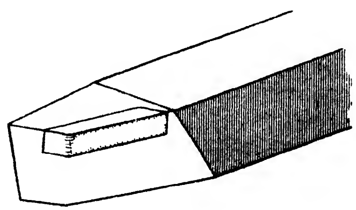


Fig. 14. Carbide Tool with Negative Back Rake of 5 to 10 Degrees

Side Rake. Side rake is even more important than back rake. Its first function is to direct the chip to the side of the tool post. Its second purpose is to make the tool easier to feed into the material and to equalize its side thrust. This reduces power consumption. In general, side rakes are from six to ten degrees for carbide tools, depending on the material that is to be cut. For very hard materials and for machining rough castings and forgings, the side rake is sometimes made negative from six to ten degrees, as shown in Fig. 15.

Relief Angles. Relief angles are provided so that the tool side or end will clear the work sufficiently, preventing it from rubbing against the work. These angles should not be so great as to weaken the tool.

A relief angle of from three to six degrees is usually sufficient, save when cutting aluminum, copper, or nickel. More relief may then be needed because the metal may stick to the flank of the tool and cause a rough finish on the surface of the work. The values of the reliefs have no influence on the force on the tool nor on the power consumed. They do, however, have a relationship to the life of the tool. For turning hard metal, the relief angles should be from three to five degrees.

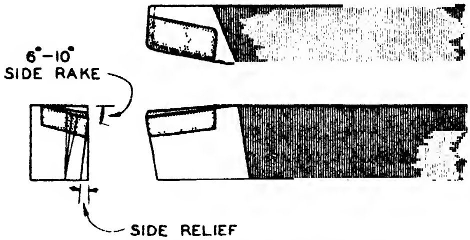


Fig. 15. Carbide Tool with Negative Side Rake of 6 to 10 Degrees, and Negative Back Rake

Approximate relief angles for most jobs are shown in Table I. These angles may be used as a guide in grinding tools for machining the materials listed. Machine operators and production men should bear in mind that, under special conditions, other angles may give more satisfactory cutting action.

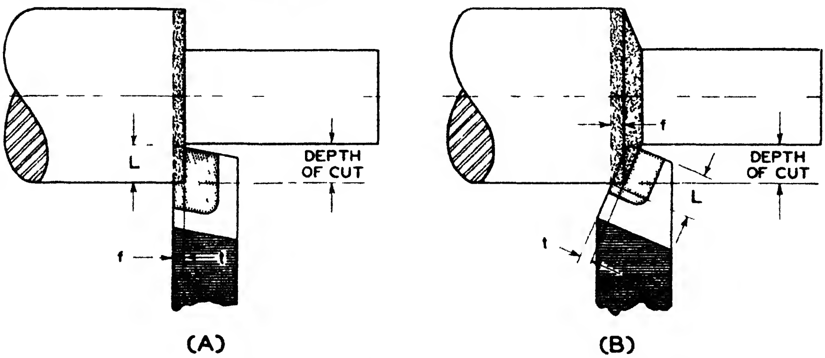


Fig. 16. The Cutting Force Is Distributed over a Longer Cutting Edge as the Side Cutting-Edge Angle Is Increased

Side Cutting-Edge Angle. The side cutting-edge angle has a significant influence on the life of the tool. The greater this angle, the longer is the life of the tool under normal conditions. The reason for this is that the distribution of the cutting force is spread over a longer cutting edge. This point is clearly illustrated in (A) and (B) of Fig. 16. Although the depth of the cut is the same in both sections of the drawing,

the length of the cut along the cutting edge of the tool is less in (A). This angle has no influence on the power consumed nor on the total force necessary to cut the metal for a given cut or feed. The size of this angle, however, does have an influence on chatter. Too great an angle will cause chatter, and chatter is highly destructive to carbide tool cutting edges. In practice, the side cutting-edge angles vary from 0° to 30°. The most commonly used angles are from 10° to 15° for roughing work and less for finishing work.

When forgings and castings are to be machined, they usually have a hard scale. For these materials, the tools should have from 0° to 10° side cutting-edge angles, because then the chip will be shorter and thicker, a condition which, on cast iron, will tend to break the hard scale ahead of the tool. A shorter and thicker chip obtained when machining steel forgings will tend to produce a greater built-up edge which will serve to protect the point of the tool. Sintered carbide tools will cut the hard scale, but at a greatly reduced speed. Therefore, to machine rough castings and forgings efficiently, small, side cutting-edge angles should be used on the tools.

End Cutting-Edge Angle. The end cutting-edge angle has no relation to the power consumed, but it does influence the life of the tool. Under normal operating conditions, this angle should be small enough to clear the work, as shown previously in (B) of Fig. 3. Normally, an angle of from 5° to 15° is sufficient, unless it is desirable to have a large radius on the nose of the tool. If such is the case, the angle should then be from 10° to 15° or even more. Existing on-the-job conditions may require a greater end cutting-edge angle because of unusual clearance conditions, but only in those cases should the angle be larger. Too great an angle leaves the tool too "pointy" and results in a tool which is not able to conduct the heat away fast enough.

Nose Radius. Nose radius has no relation to the cutting force but it does influence the life of the tool. Tools with longer nose radii leave a smoother finish when a coarse feed is used. A large radius, however, may cause chatter. Commonly used nose radii for turning tools are from 1/32" to 1/8". Where the tool is required to produce a sharp corner, there cannot be any radius on it. However, since designers of machine parts seldom call for sharp corner finish, this is not often the case.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points in the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What is meant by an intermittent cut?
2. What is meant by a negative back rake tool? For what work is the negative rake angle tool used?

3. What is meant by the cutting angle of the tool?
4. What effect, if any, has the side cutting-edge angle on the power required to make the cut?
5. What is the reason for a side cutting-edge angle?
6. How large should the end cutting-edge angle be?
7. What effect has the back rake angle on the power consumed?
8. State the meaning of the "machinability of a tool" and explain the factors which influence this quality.
9. By what methods or ways may the sintered carbide tool fail?
10. Define the continuous chip with a built-up edge, and the continuous chip without the built-up edge.
11. In machining soft, ductile metals such as soft steels, how can the built-up edge be reduced to the minimum?
12. Sketch a cemented carbide tool suitable for use in machining plastics.
13. What effect has the built-up edge on the finish produced in cutting?
14. Why is a broken chip produced in machining cast iron?
15. What effect does the use of a lubricant have in cutting such material?
16. How does cratering shorten tool life?
17. How much negative back rake is usually given a tool for the turning of rough forgings?
18. How much negative back rake is usually given a tool for shaper and planer work?
19. What are the two chief functions of side rake?
20. Why is more relief required in the tool for cutting aluminum?

REFERENCES

Begeman, M. L. Manufacturing Processes. New York: John Wiley and Sons, Inc., 1942.

Boston, O. W. Metal Processing. New York: John Wiley and Sons. Inc., 1941.

Carboloy Company. Engineering Bulletin GT-133-R. March, 1943.

Cincinnati Milling Machine Co. "Milling Machine Practice."

McCormick, R. H. (Vascoloy-Ramet Corporation). "Critical Speeds for Carbides," American Machinist, October 29, 1941.

Kennametal, Inc. "Steel and Metal Cutting Tools and Blanks."

Vascoloy-Ramet. "Tantalum-Tungsten Carbide Tools and Blanks."

CHAPTER IV

Tipping a Carbide Tool

Practicality of Tool Tipping in the Shop. While carbide-tipped tools are available in many forms and for many purposes, complete and ready for use, most users prefer to buy the carbide inserts, "blanks," or tips, then buy or make their own shanks, bars, or cutters, and assemble the components themselves. This is especially true if any reasonably large quantity of such tools are to be used. These users also have developed methods for retipping the tools as the carbide wears, thus again avoiding the expense of continually buying complete tools. Since the tipping of single-pointed or single-edge carbide tools has become such a widespread practice, this chapter will be devoted to the tipping with sintered carbide of tools for turning, parting, forming, shaping, planing, and other such applications. The methods are much the same, whatever the purpose of the tool.

Preparing the Shank. There are four major steps in the making of a single-pointed cutting tool. Each of these steps should be carefully followed to get a tool that will give the best service. These steps are the shaping of the tool shank, cutting the recess in the shank to receive the tool blank, the brazing of the tool blank to the shank, and the grinding of the brazed tool.

A typical shank shaped for a turning tool is illustrated in Fig. 1. The shank is first cut to the desired length from a cold-drawn bar of the required cross section. The front and the side relief angles are then machined. Following this, the side and front cutting-edge angles are cut, and finally, the side rake and the back rake angles are machined. This is usually done by milling. The relief angles on the shank are milled from two to four degrees more than the specified primary relief on the tool, so that after brazing, it is only the carbide tip which needs to be ground. This practice speeds up grinding and saves wear of the grinding wheels.

The machining of shanks can be done by any of three methods: milling, shaping, or grinding. Of the three, milling is by far the most satisfactory, particularly when production calls for a number of tool shanks. Milling of the side rake angle on a shank, using a shell-end face mill, is shown in Fig. 2.

Milling the Recess. After the angles on the shank are machined, the recess for the tool blank is milled. The most commonly used recesses are shown in (A) through (D) of Fig. 3. The rabbet recess

shown in (A) of Fig. 3 is used for grooving, cutting-off tools, and for round- and pointed-nose tools on which the tip extends across the shank width. It is also used for side-cutting tools such as that shown in Fig. 4. This tool may be used for turning where the cuts are light and the side thrust consequently small, and where the temperature of the tool during cutting does not rise too high. The straight recess can be easily produced by milling, shaping, or grinding.

When support for the blank against side thrust is necessary, a recess such as shown in (B) of Fig. 3 may be used. This recess can be machined with a side mill. The side-mill recess of the type shown in (D) of Fig. 3 is used for both right- and left-hand turning tools. This kind of recess can be machined with a side mill or may be done in the shaper.

The most widely used recess, however, is shown in (C) of Fig. 3. It provides for support of the carbide blank against both side and end thrust. It affords maximum shank support to the tool blank and resists

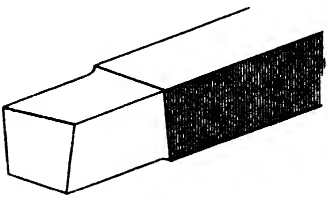


Fig. 1. Tool Shank with End Shaped for Forming Correct Cutting Angles

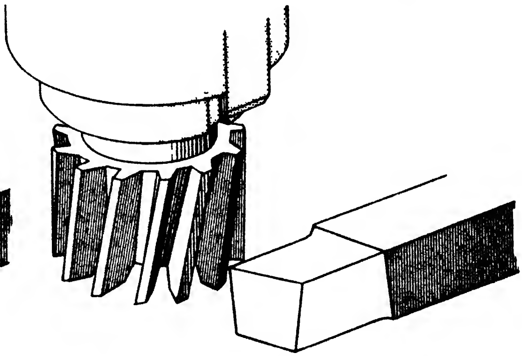


Fig. 2. Milling the Side Rake Angle on a Tool by Means of a Shell-End Face Mill

the tangential, radial, and side thrust forces. An end mill must be used to produce this type of recess.

It is of utmost importance that the recess be flat and without any "steps." The surface need not be particularly smooth, but there should be sufficient surface contact to afford ample support to the carbide blank. Care should be exercised in milling in order to prevent the cutter from remaining at any one point until it cuts deeper there than in other places. Any unevenness in the recess, particularly in the base, may cause the carbide tip to crack in service. The recess should be free from burrs or "bruises," since these will cause poor brazes, and, in addition, will contribute to the cracking of the tool in service.

Materials for Shanks. Tool shanks are made of a good grade of machine steel, usually cold-drawn. The carbon content of tool shanks varies from 0.45 to 0.95 per cent. Some users of carbide tools prefer

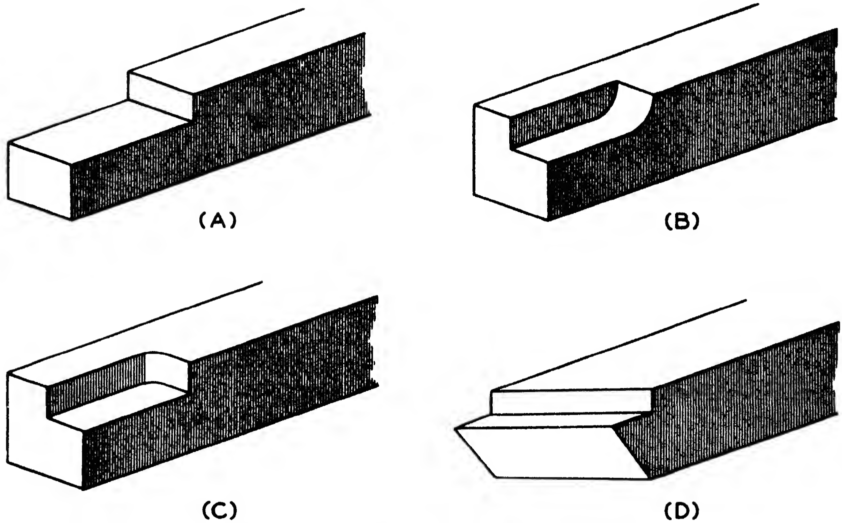


Fig. 3. The Rabbet Recess (A); a Recess Milled to Give Side Support (B); a Recess That Gives Support against Both Side and End Thrust (C); and a Recess That Gives Support against End Thrust Only (D)

alloy steel, and others even high-speed steel. Common carbon steel with a carbon content of 0.65 to 0.95 per cent, however, gives entirely satisfactory service when the designer gives due consideration to the rigidity of the shank. By "rigidity" is meant the resistance of a material against deflection.

Alloy steels, including high-speed steel, usually have greater strength than the carbon steels, but their elastic properties are about the same. Therefore, the rigidity of carbon and alloy steel units of identical size is practically the same, and little or nothing is gained by the substitution of the alloy steel. Tool shanks made of Meehanite, a cast ferrous alloy, are also used. When they are properly made and tipped, they give a very good performance.

Brazing the Tip to the Shank. The third step in the making of a sintered carbide cutting tool is the brazing of the carbide blank or tip to the shank. This, also, should be done carefully or poor adhesion may result with consequent tool failure. In general, there are three methods of brazing tips to shanks. These are: torch brazing (using an oxyacetylene welding torch), furnace brazing (either gas or electric), and induction coil heating.

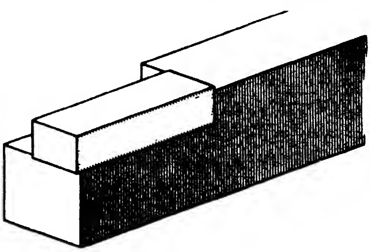


Fig. 4. A Side-Cutting Tool with Rabbet Recess

Each of these methods possesses

certain advantages and disadvantages which will be discussed step by step. The application of one method or the other is a matter of economy in production, usually dictated by the number of tools to be produced and the equipment available.

Brazing Defined. Brazing, for the uninitiated, is a general term applied to the joining, without melting, of iron, cast iron, malleable cast iron, steel, brass, bronze, copper, nickel, and many other metals in similar and dissimilar combination by means of a nonferrous "filler" material. It is also defined as a series of metal-joining operations or processes in which the filler material is a nonferrous metal or alloy, the melting temperature of which is more than 1000° F., but less than that of the metals to be joined together.

The secret to successful brazing is the proper use of natural capillary attraction or, as the shop men call it, the proper "wetting" of the material to be brazed by the molten brazing material.

When the pieces to be united are properly prepared—thoroughly cleaned, fluxed, and heated to a temperature that causes the brazing material to melt and flow—a brazed joint is possible in which the metals will be strongly fastened. It is well known that a properly brazed joint offers a stronger fastening than bolting, riveting, or even spot welding. This is because the entire contact surfaces of the two parts which have been united form a joint which has a much larger area than that fastened.

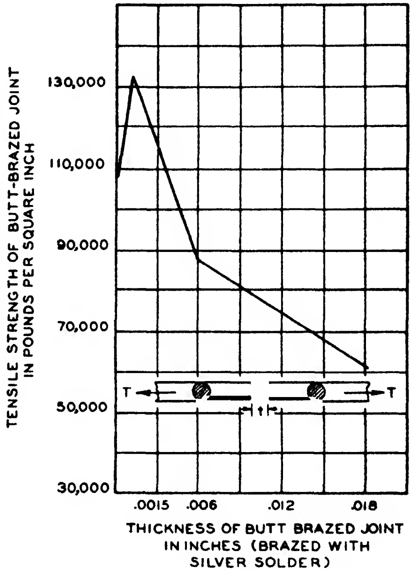


Fig. 5. Tensile Strength of a Brazed Joint, Based on the Thickness of the Braze

If the parts to be brazed are designed with a proper fit or clearance, the brazed joint may be even stronger than the material joined. This will depend on the exactness of the surfaces to be brazed, the cleanliness of the materials, the brazing material, the temperature at which the brazing is done, and the time consumed. Fig. 5 is a graph which shows the strength and thickness relationship of a brazed joint made with silver alloy.

Torch Brazing of Tools. When only a small number of tools are to be made up, an oxyacetylene torch will be found to be the most satisfactory and economical method of brazing. The average shop is equipped for this work, since most of them have welding equipment. The method calls for an operator who is skilled in the use of a gas torch.

He must be one who will exercise care in the uniform heating of the tool shank and the tip, since localized overheating may crack the tip if the torch is allowed to play too long on one spot.

It is essential for torch brazing that the metals to be brazed are clean and free from grease, scale, or any other foreign material which may interfere with the capillary flow ("wetting" action) of the brazing metal. Good brazing technique calls especially for a thorough cleaning of the parts to be joined. This cleaning may be mechanical or chemical. Mechanical cleaning may be done by sand blasting. Chemical cleaning may be done with degreasing equipment or by dipping the parts in a suitable solvent. Chemical cleaning also is done by dipping the shank and the blank in hydrochloric acid for a few moments to clean off any rust, oil, or dirt, and then in carbon tetrachloride to remove all traces of acid and grease. Besides cleanliness, it is necessary that the edges of the recess in the shank be evenly milled and free from burrs and fins. The carbide tip should fit tightly, because only a thin film of brazing material is needed to make a sound joint.

Common Metals and Fluxes. The most commonly used metals for brazing are copper in sheet form, Tobin bronze, Chamet, or Castolin rod brazing alloys, and Handy and Harman Easy-Flo #3 alloy.

Copper has a high melting temperature, about 1980° F. For this reason it is not an easy material to use for torch brazing. Copper also oxidizes easily, and unless borax is used as a flux in sufficient quantity, poor brazes will result. However, copper, with certain limitations, is suitable for furnace brazing. Tobin bronze melts at about 1600° F. It produces good but not necessarily consistent brazes. This is true of most of the bronze alloys. Handy and Harman Easy-Flo #3 alloy melts at about 1275° F., and results in quite uniform brazes. Moreover, blanks brazed with Easy-Flo #3 can be easily removed from the shank and replaced.

Borax, Airco-Marvel, Oxweld-Brazo, Fluxine, or almost any of the other fluxes prepared by reputable manufacturers are used for brazing with bronze rod. Handy and Harman paste flux usually is preferred for use with Easy Flo #3.

How Brazing Is Done. The tipping job varies slightly with each brazing medium used. As a means of presenting a clear picture of these processes, two different, complete operations will be described and followed through, step by step.

Brazing with Tobin bronze or Easy-Flo #3 consists in, first, the selection of a proper torch tip for the job at hand. The proper tip to use is largely a matter of the experience of the operator but, in general, the greater the size of the tool to be brazed, the larger the tip should be.

A thoroughly cleaned shank, free from burrs, is then placed in the vise as shown in (A) of Fig. 6, the flux applied, the torch adjusted to a nonoxidizing flame as shown in (B) of Fig. 6, and the shank preheated as in (A) of Fig. 6.

When preheating of the shank with the nonoxidizing flame has reached

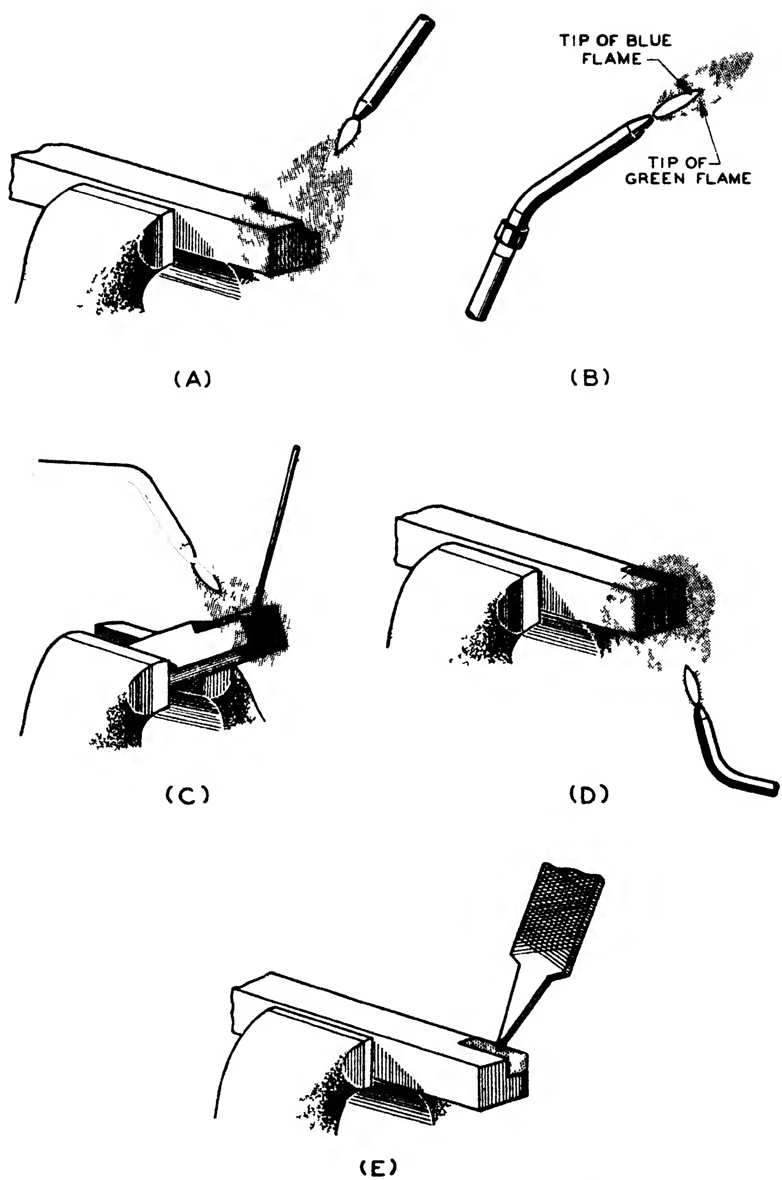


Fig. 6. Sequence of Heat Application Steps in Torch Brazing of Single-Point Tools

the stage where the flux melts and runs free, the recess is tinned with brazing material as shown in (C) of Fig. 6. This is done by touching a 1/16'' round rod of bronze or Easy-Flo to the recess of the shank. The carbide tip, previously fluxed and sometimes tinned, is now placed in the recess with tongs or pliers.

The whole end of the tool shank with the tip is now heated. This heat is applied to the underside of the shank with the torch, as shown in (D) of Fig. 6, until the flux and the bronze are melted again. The carbide tip is then heated. During this operation, care must be exercised that the torch is not allowed to remain at any one point too long. It should be moved about so that the tip and the tool end of the shank heat evenly. If the tip is not heated evenly, it may crack because of the difference in its expansion rate.

Once the assembly has been heated to the proper temperature, the blank is pressed firmly into the recess as shown in (E) of Fig. 6, and held there for a few seconds until the silver or bronze has solidified. This procedure squeezes out the excess brazing material and assures a tight joint. Finally, the flux scale and excess brazing material are removed from the tool shank and the point is ground. Flux scale can be removed with a wire brush and hot water. Excess brazing metal can be removed on an emery cloth wheel, but care should be exercised that the tool is not overheated.

Brazing with Easy-Flo #3 alloy or silver solder rolled into sheets is done in the following manner. All surfaces to be joined are cleaned with carbon tetrachloride. This fluid may be applied with a brush as shown in (A) of Fig. 7, and serves to clean the recess preparatory to brazing. Pieces of sheet alloy from .003 to .005 in thickness are then cut slightly larger than the shank recess as shown in (B) of Fig. 7.

Following this, the recess surface is coated with flux as shown in (C) of Fig. 7, and the strip of Easy-Flo #3 placed in the recess. The top part of the Easy-Flo is coated with flux, and the carbide tip or blank placed in position.

The oxyacetylene torch is now adjusted to a nonoxidizing flame, as was shown in (B) of Fig. 6. The tool assembly is held in the vise and heat applied to it as in (D) of Fig. 7. The shank should be heated first, applying the flame to the bottom, some distance from the point, moving the torch around from one side to the other so that no part of the tool becomes overheated. After the proper brazing temperature has been reached (which is when the brazing metal melts and the end of the shank and the blank become a dull, cherry-red color), the blank is moved into position with a piece of rod or the tang end of an old file. This operation is shown in (E) of Fig. 7.

When the Easy-Flo has completely melted, the flame is removed from the tool and the blank lightly pressed down in the shank recess in the same manner as shown in (E) of Fig. 6. The blank is held there until the brazing material solidifies. This will be at about 1200° F. The pressing down on the blank should squeeze the excess of brazing mate-

rial from the cavities, assuring a tight brazing union. Scale, flux, and excess brazing alloy are cleaned off as explained in the description of brazing with Tobin bronze.

Low-Temperature Welding. Several eutectic alloys have been developed for low-temperature "welding," which, in reality, is the

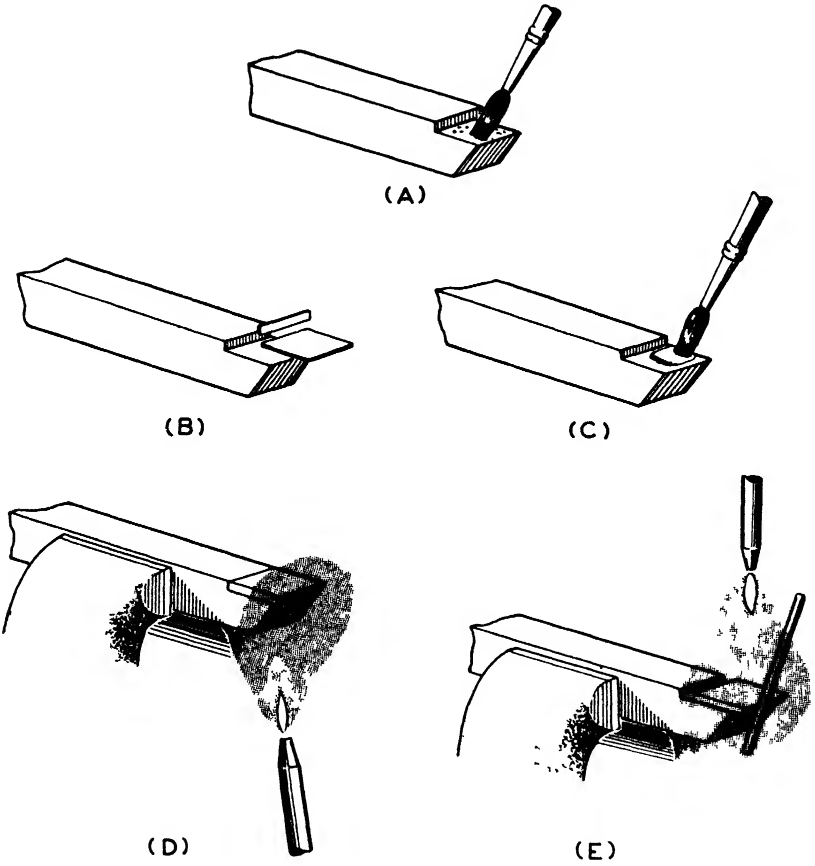


Fig. 7. Sequence of All Steps in Torch Brazing of Single-Point Tools

same process as torch brazing. Fig. 8 shows two tungsten carbide tools which have been "welded" with such an alloy, the melting temperature of which is about 1100° F. This method can be applied not only to the brazing of such sintered carbides as Carboloy, Firthite, Vascoloy-Ramet, or Kennametal, but to the cast alloys, such as Stellite and Tantung, as well.

The advantage of brazing at low temperature is that the possibilities of shank and tool blank distortion are minimized, thus reducing the

chances of tool tip cracking as the result of brazing strains. The disadvantage is that alloys melting at low temperature may not have enough holding power when the tool reaches a temperature much higher than 1000° F. under operating conditions.

"Welding" procedure with these eutectic welding alloys is similar to that with torch brazing. The surfaces to be brazed are dipped in hydrochloric acid for a few moments. This removes rust, grease, oil, or dirt. These surfaces are then cleaned in carbon tetrachloride to

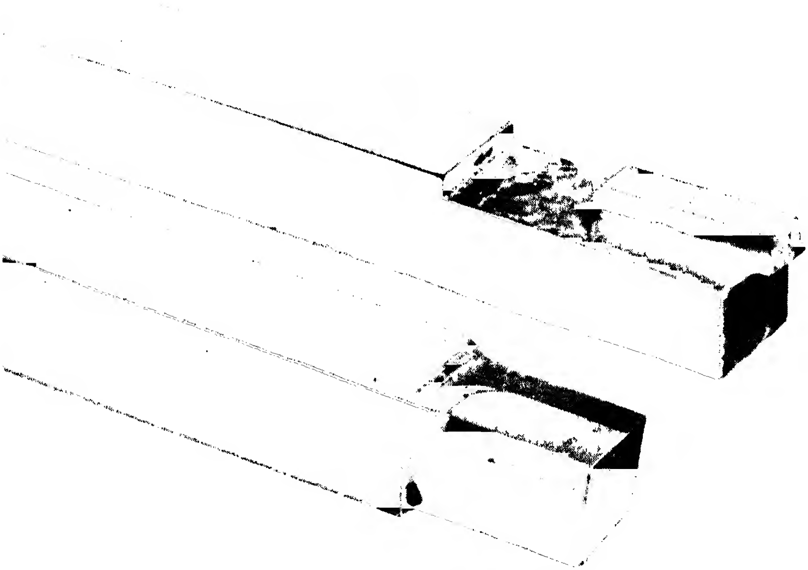


Fig. 8. Tool "Welded" with Low Temperature Brazing Alloy
Courtesy of Futectic Welding Alloys Co

remove all traces of the acid. Following this, the surfaces to be brazed are coated with a suitable flux. The pieces of eutectic alloy are placed in the recess of the shank, after which the blank is laid in position.

The oxyacetylene torch, adjusted as in (B) of Fig. 6, is used to heat the shank from the bottom and the sides until a cherry-red color is reached. Heat is then applied to the top of the blank and to the sides of the shank until the alloy is fully melted. At this point heating is discontinued and pressure is exerted on the blank with the tang end of a file until the braze has solidified.

The completed tool is then cooled slowly either in air, powdered mica, powdered graphite, or lime. The excess material is then cleaned from the shank and the tool ground to the required point.

Furnace Brazing of Tools. When only a small number of tools are to be assembled, torch brazing is economical and entirely satis-

factory. Where sintered carbide tools are used on a much larger scale, furnace brazing is a better choice.

Small quantities of such tools may be brazed in a gas furnace of the type that is completely muffled. The method consists of heating the previously prepared shank and tool blank together with the brazing material, all properly fluxed, to temperatures at which the brazing material will melt and "wet" the surfaces to be brazed. After this temperature is reached, the tool is partially withdrawn from the furnace and the blank pressed down into place while it is still at the threshold of the furnace. Successful operation of this method requires a nonoxidizing or neutral atmosphere in the furnace. Otherwise, oxidation of the materials may prevent proper brazing action.

In common furnace brazing, the step-by-step procedure involves cleaning the shank and the blank thoroughly, either chemically or mechanically. Some brazing operators prefer to sandblast the surface to be brazed after the degreasing operation. This provides a satin-surface finish which is conducive to good "wetting" by the brazing material. Next, flux is applied to the surfaces to be brazed in a quantity just sufficient to float off the impurities. An excess of flux should be avoided.

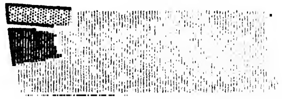


Fig. 9. Misaligned Tip on an Improperly Brazed Tool

The shank and the blank are then assembled with brazing material between them. If the shape of the tip is such that it will not stay on the shank properly, it may be necessary to tie the parts together with a wire or asbestos string. Following this, flux is applied to the top of the assembly and is allowed a few minutes in which to dry. Additional brazing material also may be placed on top of the assembly. The parts are then placed in the muffle of the furnace, which is set to operate at a temperature somewhat higher than is needed to melt the brazing material.

When the brazing material has melted, the tool is pulled out. The blank is adjusted in position and is pressed firmly into the recess. It is held there until the brazing material solidifies. This will eliminate misaligned tips or the kind of poor brazes shown in Fig. 9. The brazed tool is slowly cooled to avoid brazing strains. This is best done in crushed mica, lime, or graphite. Following the cooling of the tool, it is cleaned of flux glaze and of excess brazing material, and is finally ground.

Atmosphere Electric Furnaces. Electrically heated furnaces are playing an increasingly large part in the production of carbide tools. Economy, increased rate of production, and the high quality of the finished product, achieved through the use of controlled, protective atmosphere in the furnace, are the chief advantages of this method of heating. There are three types of furnaces commonly used for this work. They are known as the plain muffle type, the pusher type, and the belt-conveyor type. The latter especially is used where tools are produced on a large scale.

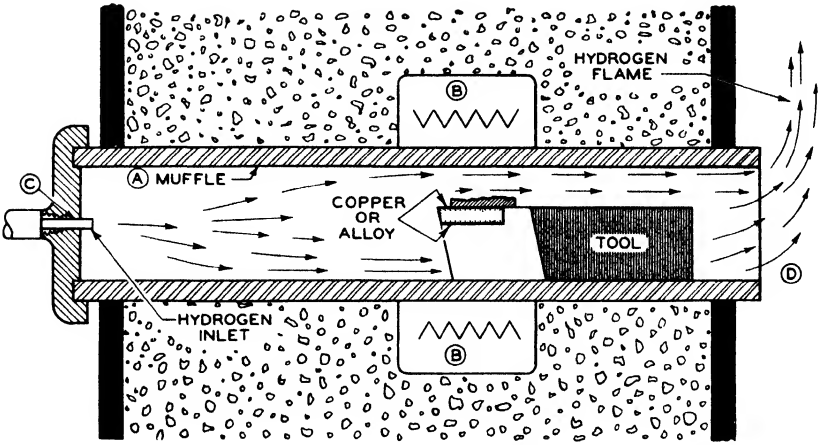


Fig. 10. Principle of the Electric Furnace Having a Hydrogen Atmosphere

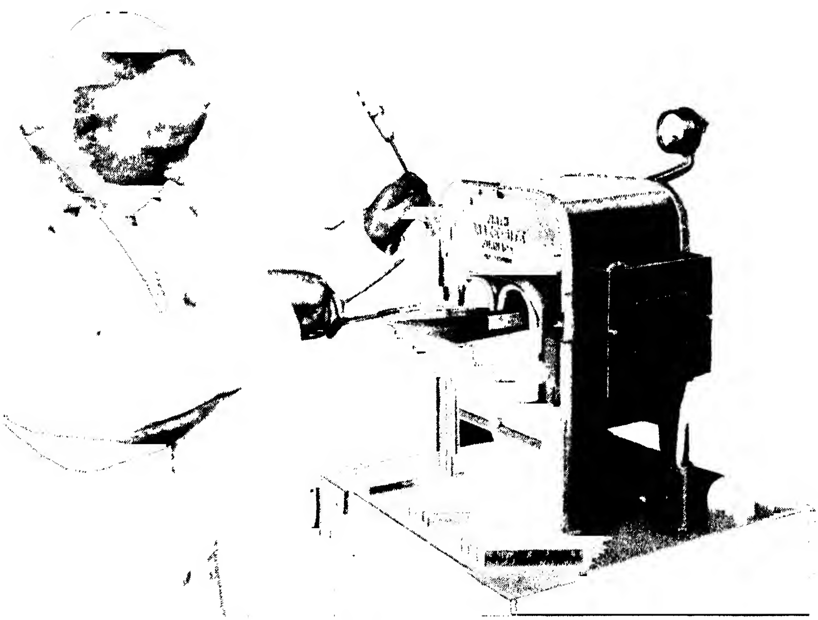


Fig. 11. Electric Furnace with Controlled Atmosphere

Courtesy of Firth-Sterling Steel Co

The principle of the electrically heated furnace with controlled atmosphere is illustrated in Fig. 10. The furnace consists of a muffle, A, made of refractory material, and a heating zone with electric heating elements, B. Nonoxidizing, moistureless hydrogen gas is passed through the muffle from inlet C and out of an opening or door at D, where it burns with a small flame. In larger installations, the furnace is equipped with a reasonably tight door, and the hydrogen oozes out through the small crevices around it.

When the door is opened, a curtain of fire at the entrance to the furnace is lighted automatically or manually. This curtain prevents air from rushing into the furnace.

Fig. 11 shows a small, muffle-type electric furnace designed for brazing with copper or any other brazing material. This furnace is used when the number of tools to be brazed is somewhat greater than can be easily handled with the oxyacetylene torch method. Its chief advantages are the uniformity of heating and the controlled, nonoxidizing atmosphere, both of which are essential in brazing with copper and highly desirable in brazing with other materials.

The brazing steps involved in the use of this equipment consist first in the preparation of the furnace by bringing its heat up to 1550°-1600° when brazing with Easy-Flo, or to 2000°-2100° if using copper strips. While the furnace temperature is being raised, the tool shank and the blank should be cleaned of all rust, oil, grease, and dirt. The surfaces to be brazed should then be lightly sandblasted, after which they should be coated with a sufficient amount of flux. A piece of braze material is placed in the recess, followed by the blank, then by a second piece of braze material on top of the blank, as shown in Fig. 10. The assembly is then placed in the furnace with the tip of the tool in the heating zone.

When the tool point has reached the temperature at which the brazing material is completely melted, the tool is partially withdrawn from the furnace. At the furnace door, the blank is adjusted and pressed into the recess. It is held in place with a suitable poker or an old file until the braze has solidified. The pressing squeezes out excess brazing material and assures a sound braze. Following this, the tool is cooled to room temperature, preferably slowly, in crushed mica, or in lime. It is then cleaned of excess brazing material and flux glaze, and is ground according to the job it is to do.

High-Production Furnace Brazing. When a large number of tools are to be produced and the production is continuous, the brazing methods described in the preceding pages are not efficient enough. To meet this larger demand for brazed tools, electrically heated furnaces with controlled atmosphere are used. Furnaces such as those shown in Figs. 10 and 11 have somewhat limited capacities as to the number of tools that can be heated at any one time. Consequently, they limit production.

For larger output, then, a pusher-type furnace, shown in Fig. 12, is

used. Where tool production requirements call for continuous operation, a belt-conveyor type of electric furnace with controlled atmosphere is preferred. This furnace is shown in Fig. 13.

The pusher-type furnace consists of a rectangular shell which is well insulated and provided with a heating chamber, A, which has a flat

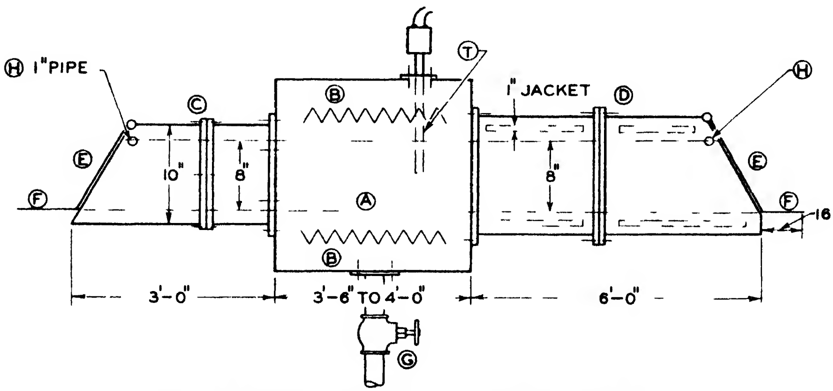


Fig. 12. Principle of the Push-through Type of Furnace Used in the Brazing of Cutting Tools

bottom of the rail type. Surrounding the heating chamber are the electric heating elements, B, which supply the heat needed for brazing. Attached to the furnace, thus forming an integral part of it, are rectangular tube sections C and D. These sections form a preheating chamber at C, and a cooling chamber at D. The lengths of these chambers may vary. The preheating chamber is usually from two to four feet in length, and the cooling chamber is from four to six feet or even more in length. The inside dimensions of the furnace and of the tubes depend on the size and capacity of the furnace. The heating chamber and tubes shown in Fig. 12 represent an average-sized furnace. The main section of the

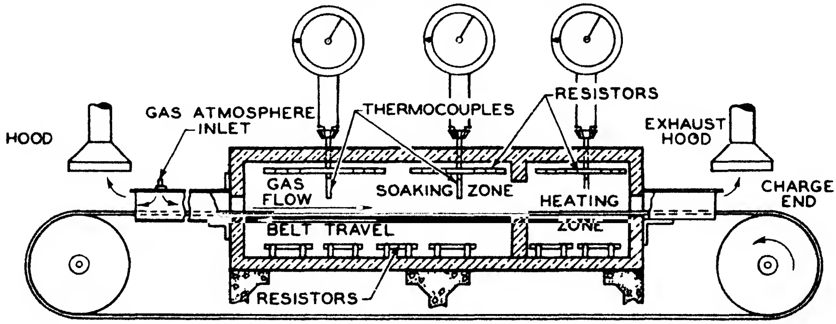


Fig. 13. The Belt-Conveyor Type of Electric, Controlled Atmosphere Furnace

furnace is usually from three to four feet in length. The cooling section is jacketed for the circulation of water. The preheater and the cooler are provided with close-fitting doors of the lifting or sliding type shown at E. The closeness of the fit holds the special atmosphere in the furnace. Extensions at F are the loading and unloading platforms which are placed level with the floor of the furnace so as to facilitate the sliding in of the tools to be brazed. A suitable pushing tool for this purpose is illustrated in Fig. 14.

Controlled atmosphere gas enters the furnace through the connection at G, and would escape through the doors if they were left open. At both

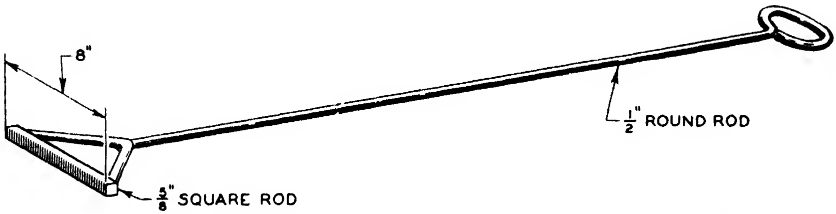


Fig. 14. A Hand Pushing Tool for Shoving Tools through the Furnace

doors at H, inside the cooling and preheating chambers, there are burners, connected to the city gas supply line, consisting of pipes with rows of small holes. The operation of these burners is so arranged that a curtain of fire is started at the entrance to the furnace as soon as the door is opened. The ignition of the curtain of fire is either manual or automatic. In the latter case the gas is ignited by a small pilot gaslight which burns continuously, close to the junction of the door and the tube. The function of this curtain of fire is to prevent the air from rushing into the furnace when the door is opened. This is necessary because any air present in the furnace during heating would cause scaling of the tool shanks or bodies by oxidation.

Single-point tools are prepared for brazing in this type of furnace in the usual manner. The tool shanks and inserts are fluxed and assembled with a brazing alloy, then placed on graphite plates or trays as shown in

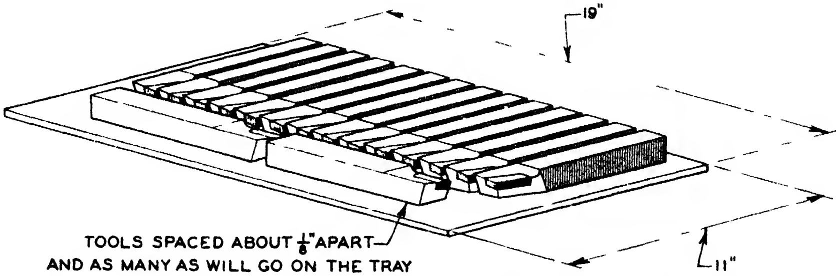


Fig. 15. Graphite Boats (or Plates or Trays) Used to Carry the Tools through the Furnace

Fig. 15. These trays are pushed carefully into the furnace from platform F. Other trays are placed in succession behind the first, thus pushing the front trays into the heating chamber of the furnace. Once the tools have been heated to brazing temperature, they move into the cooler and are finally pulled out at the opening to the cooler. During this operation the furnace remains closed except for the very short periods required for placing the trays inside and removing them when the cycle of preheating-brazing-cooling has been completed.

Exhaust hoods are placed over the entrance and removal doors to vent the unburned gases from the working room. The temperature is recorded through a thermocouple placed in the furnace and indicated by T in Fig. 12. This thermocouple is connected to a temperature indicating instrument and temperature control apparatus or instruments.

It should be noted that when the tools are taken from the furnace, they are finished so far as brazing is concerned. The brazing in the furnace is accomplished through the capillary action of the brazing material and no wiping in or pressing of the tool blank into the recess of the tool shank is necessary as is the case in torch or gas furnace brazing. It will be necessary for some tools to tie the blank in its correct position in the shank recess, using heat resistant wire or asbestos string. This will prevent the blank from falling out or becoming misaligned in the tool shank.

Furnace Brazing Temperatures. The temperature range of brazing is from 1200° to 2050°, or even 2100° F. The lower heat is used where the brazing media consist of silver alloys with copper, cadmium and zinc, or combinations of copper and phosphorus. The higher temperatures are used for brazing with copper and bronze.

When brazing steel shanks and sintered carbide blanks with copper, the furnace is operated to 2050° F., plus or minus 15°. This temperature will give a good joint, provided the brazing surfaces are well prepared. For brazing with silver solder such as Easy-Flo, a temperature of 1600° F. is usually employed, although the brazing material melts at a much lower temperature. Extremely high brazing temperatures should be avoided since they bring about grain growth in the steel shanks, resulting in weakened tools.

Controlled Atmosphere. It was previously stated that a controlled furnace atmosphere is essential for the best results in furnace brazing. There are several kinds of controlled atmospheres. Mentioned heretofore has been hydrogen, which is nonoxidizing but slightly reducing. That is, it will not form oxides but may slightly decarburize a steel surface. Another atmosphere commonly used is Endogas, a fuel gas very rich in methane that has been partially cracked. This gas is used for brazing operations to prevent decarburization when medium- and high-carbon-steel shanks are involved. Amnogas, which is dissociated ammonia formed by cracking anhydrous ammonia, is another frequently used atmosphere. It is high in hydrogen. Still another is Exogas, a mixture of either manufactured or natural gas and air in the proportions

necessary to obtain the desired atmosphere. Monogas, similar to Exogas, is produced in the same sort of generator. Table I gives pertinent data for these typical, controlled atmospheres.

There are other analyses of gases besides those listed in Table I which may be classed as lean, medium, and rich Amnogas, Endogas, or Exogas. They all have uses in certain particular or special brazing problems.

The chief advantage of the controlled atmosphere in the production of brazes is the cleanliness and uniformity of the work that comes from the furnace. Frequently, the only cleaning that is necessary after the brazing is the removal of the flux glaze. This obviously results in economy in production as well as better and stronger brazes.

TABLE I. TYPICAL CONTROLLED ATMOSPHERE

Name of Gas	Air to Gas Ratio	CHEMICAL COMPOSITION						Kind of Atmosphere
		N_2	CO	CO_2	H_2	CH_4	O_2	
Amnogas	0-1	25.0	0.0	0.0	75.0	0.0	0.0	Reducing
Burned Amnogas	1.8-1	99.0	0.0	0.0	1.0	0.0	0.0	Inert
Endogas	2.75-1	41.7	19.0	0.0	38.0	1.3	0.0	Reducing
Exogas	6-1	69.0	10.0	5.0	15.0	1.0	0.0	Reducing
Monogas	10-1	98.0	0.0	0.0	1.0	0.0	0.0	Inert

Belt-Conveyor Furnaces. For exceptionally large quantity production of sintered carbide tools, a continuous, belt-conveyor type furnace is used such as that shown in Fig. 13. The prepared tools may be placed directly on the belt of the conveyor or on trays which are then placed on the conveyor. The link-type belt is made of high-chromium and high-nickel steel, or other heat-resisting alloy. The chief advantages of this process are the continuity of operation, ease of handling, and uniformity. The disadvantages are the maintenance cost of the conveyor and its operating mechanism and a greater loss of the atmosphere gas.

The principle of operation of this furnace is identical to that of the pusher type except that the movement of tools through the belt-conveyor furnace is continuous and automatic.

Precautions To Be Observed. Hydrogen itself is a harmless gas, but, mixed with air, it becomes a powerful and dangerous explosive. Cracked gas, used as an atmosphere-control medium and containing carbon monoxide and hydrogen, is also explosive and should be handled with care. When starting any controlled atmosphere furnace, the following precautions should be observed:

When the furnace is not in use and standing cold, the doors should be left open. Before starting the heating elements, the furnace should be checked to make certain no dangerous mixture of gas and air is present.

After the absence of explosive mixtures of gas has been determined, electric power is turned on and the furnace is brought up to the desired temperature, the doors remaining open. Hydrogen or cracked gas is now admitted.

The furnace doors should be kept open while the hydrogen or cracked gas is slowly valved into the furnace. The incoming gas will ignite immediately on entering the furnace and will burn without exploding or blowing out with the flame toward the doors. After the gas has burned for two minutes, the doors are closed and the city gas pilots are lit. The hydrogen valve is adjusted so the flame is not more than two inches above the edge of the door, or the bleeder pipe if one is used. Otherwise, there will be an unnecessary wastage of gas.

One should never stand in line with the door openings when putting the furnace in operation. A noninflammable face shield should be worn when it becomes necessary to look directly into the furnace in starting, operating, or shutting it off. Pilot lights should be burning while the furnace is in operation, since their purpose is to ignite escaping gases. There should never be any smoking or lighting of matches near the furnace or around the hydrogen lines when the furnace is not in operation.

High-Frequency Induction. In furnace brazing, the whole shank or tool body is heated, together with the assembled tool blank or tip and the brazing alloy. In such work the shank is subject to considerable expansion on heating, and contraction on cooling. The latter sometimes leaves serious strains in the shank and in the tool tip itself. Expansion rates of the tool shank and the carbide blank are not the same. Consequently, considerable strain is set up in the braze and the carbide tip. Occasionally, this strain is so great that the latter will crack. This is known as a brazing crack.

In order to avoid this cracking of tips, other methods of brazing are employed so as to localize the heating, confining it solely to the point being brazed. It will be noted that torch brazing was a step in this direction, but torch brazing is slow, therefore uneconomical where high production schedules are demanded. A faster system of localized heating by means of high-frequency alternating current is often used, the heating being done by current induced in the part to be brazed.

Induction heating is a means of raising the temperature of a metallic part by the transfer to it of high-frequency electric energy from a current-carrying conductor called a heating coil. This coil sets up a field of magnetic flux which energizes the piece of work in such a way that current is caused to flow around its surface. The resistance of the work to this flow causes the heating action. The temperature of the metallic part thus is raised by the electrical generation of heat within the material. The unit to be heated is in no way a part of the electrical circuit.

To generate heat in a given part, it is necessary to have a current-carrying conductor in a form of a coil to surround the work as shown in

(A) of Fig. 16. The high-frequency current enters at terminal X and passes around the coil as shown by the arrows, leaving by the second terminal at Y. The current sets up a field of magnetic flux which flows around the surface of the work, as shown in (B) of Fig. 16, in a direction opposite to that of the current in the coil. This heats the surfaces of the work which are in the field.

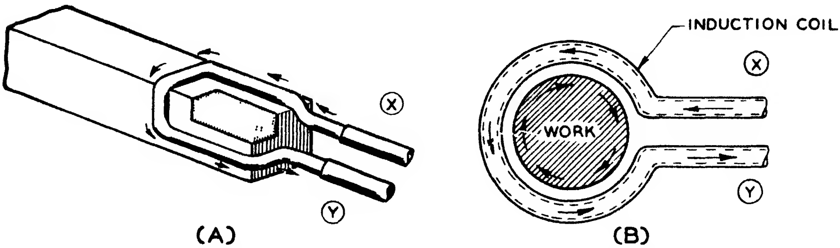


Fig. 16. Induction Coil Used for Making Braze and Diagram, Showing Current Flow

The coil is usually made of copper tubing 1/8" to 3/16" in diameter. To keep this coil from overheating and melting, water is circulated through it. All heating coils must be cooled, and the best cooling medium yet found is cold water.

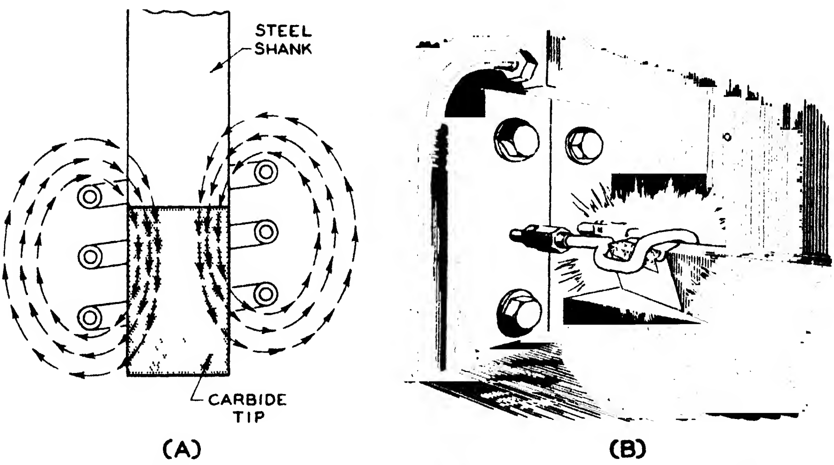


Fig. 17. Basic Principle of Induction Heating (A). The Carbide Heats Through, while the Shank Is Heated Only on the Surface To Be Brazed (B)

Basic Principles of the Method. In order to become better acquainted with the brazing of carbide tools by high-frequency induction heating, the heating cycle will be explored. Carbide is composed of finely divided particles of electrical conducting materials which are

compressed and sintered into a blank of the desired size and shape. When this material is placed in an alternating current magnetic field, such as is present in the coil illustrated in (A), Fig. 17, currents are caused to flow within the mass as shown by the arrows in the figure. The path of these induced currents is from particle to particle. The contact resistance of the material itself causes it to be heated uniformly

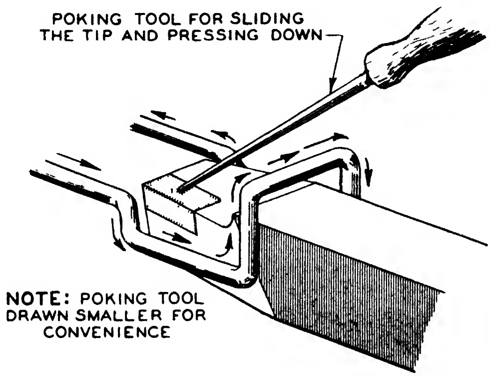


Fig. 18. A Coil Designed for Optimum Heating in a Particular Brazing Job

by the electric current induced in its own body. The electrical resistance of the carbide increases as the temperature rises, so that up to a certain point the heating effect increases at an ever-mounting rate.

The tool shank is made of steel, cast iron, or an alloy, materials possessing even greater magnetic qualities than the sintered carbides. Steel will heat more rapidly than carbide until a temperature of approximately 1300° F. is reached. At this point steel loses its magnetic properties and its temperature-rate rise will fall off. That is, it will heat much more slowly than before.

While the carbide is heated uniformly, resulting in very few if any internal stresses, the shank, on the other hand, will heat only at the surface, as is common in induction heating of such homogeneous materials as steel. How this works is shown graphically in (B) of Fig. 17. This condition is favorable, since it conserves energy in heating only the surface to be brazed and, at the same time, reduces the distortion of the shank from thermal expansion. This reduces the tendency of the carbide to crack off as a result of the steel having almost twice its thermal coefficient of expansion. The expansion of the heated surface of the steel is counteracted by the restraining force of its relatively cold mass, and the effective strain in the steel is made nearly equal to that of the carbide, resulting in very little shearing stress on the brazing alloy.

Brazing Operation. It has been shown in torch and in some kinds of furnace brazing that it may be necessary to press the carbide

blank firmly into the recess, or to skid it or puddle it into place while the brazing alloy is in a molten state. This procedure "wets" the surfaces to be brazed and assures a strong joint. To permit the same degree of easy handling of the tool in induction brazing, and to allow moving the tip after it has become thoroughly heated, the heating coil must be designed with these points in mind. Such a coil is shown in Fig. 18.

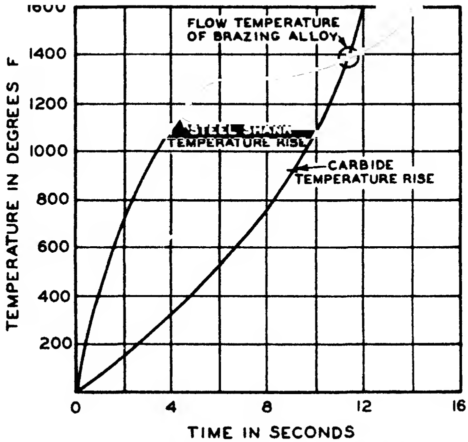


Fig. 19. This Graph Plots Temperature Rise in the Carbide Tip against That of the Steel in the Shank

This coil will place the carbide within a stronger magnetic field than the shank, so that the respective rates of temperature rise of the shank and of the tip will meet at the flow temperature of the brazing alloy. That is, they will be of the same temperature when the alloy has attained its greatest flowability. This is shown as a graph in Fig. 19. As can be seen from this graph, the heating cycle is extremely short, lasting but 12 seconds. The entire cycle, aside from the preparation of the assembly for brazing, need not take longer than 15 seconds. This is obviously much faster than torch or furnace brazing.

Normally, gravity will hold the tip in position. Where the intricacy of the tool does not lend itself to this method, binding with nichrome wire or with asbestos string usually accomplishes the desired result. The silver brazing alloy is usually used in shim form between the tip and the shank, although it may also be placed on the top of the tip. The thickness of the shim stock is determined by the amount of alloy needed in the joint. The surfaces to be brazed should be brought simultaneously to the flow temperature of the brazing alloy. Therefore, the induction coil should be of a design to accomplish this function.

Induction Heating Machines. There are several induction heating machines on the market which do an efficient job of brazing. One of these, illustrating the use of a tool-tip holding fixture, is shown

in Fig. 20. This is a Tocco 15 kilowatt motor-generator machine.

The General Electric high-frequency electronic heater is illustrated in (A) of Fig. 21. The heating cycle is accurately controlled by an automatic sequence timer. This machine is used effectively for brazing sintered carbide tools.

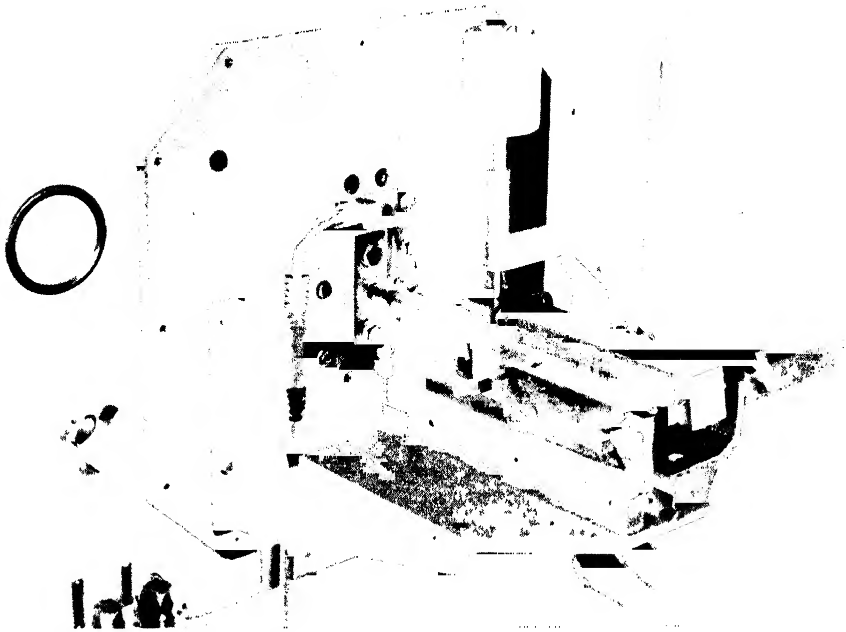


Fig. 20. The Tocco Induction Heating Machine
Courtesy of the Ohio Crankshaft Company

In (B) of Fig. 21 is shown the Ajax Electrothermic converter of the spark-gap type, another induction heater which does effective carbide brazing. This unit operates on a frequency of approximately 35 kc. (35,000 cycles per second).

A typical vacuum tube generator is illustrated in Fig. 22. This set has a rating of 20 kw output and operates at a frequency of 375 kc. The leads are shown on the side of the machine, to the right of the instrument panel.

Sandwich Brazing of Large Tips. Described earlier in the chapter was the process of brazing small carbide blanks to tool shanks with silver solder or copper having a thickness of from .003'' to .005''. It will be recalled that this material comes in sheet form, and is cut to proper size and assembled with the shank and the blank before brazing. The thickness of this brazing material gives a very close and strong braze, and at the same time provides a shearing area, or a

medium for the taking up of strains, between the carbide tip and ,oss steel shank. The strain in brazed joints is always present regardless of the brazing method used, due to the unequal contraction of the carbide tip and the shank.

Tools having carbide tips 3/4" or larger in width, thickness, or

Fig. 21B. *(Right)* The Ajax Electrothermic Converter
Courtesy of Ajax Electrothermic Co

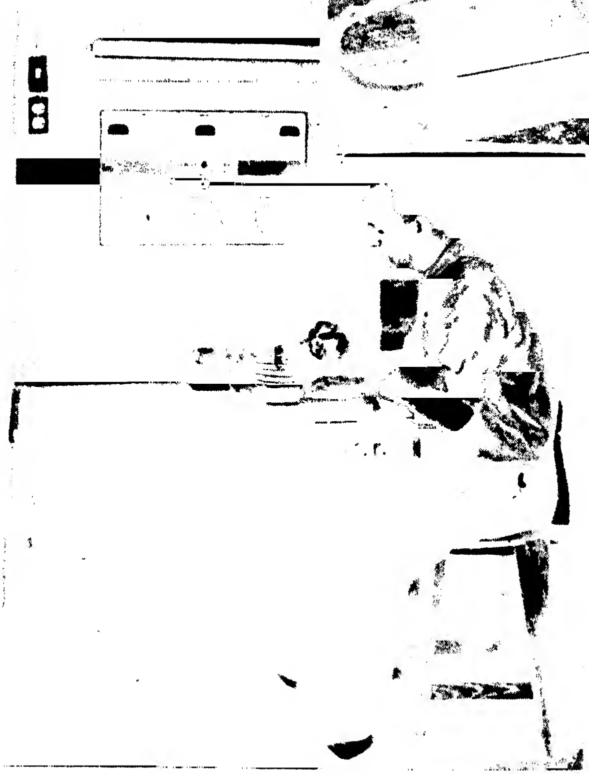


Fig. 21A. *(Left)* A High-Frequency, Electronic Type Heater
Courtesy of General Electric Company

length are more likely to crack in a brazing operation than tools with smaller tips because the amount of the strain is greater. Again, large tools are more likely to crack in grinding than the small tools, not only because of the strains set up in brazing, but also because of special grinding difficulties. Therefore, unless a special method of brazing is employed, losses by cracking may be considerable. The most widely used of these special methods is known as "sandwich" brazing.

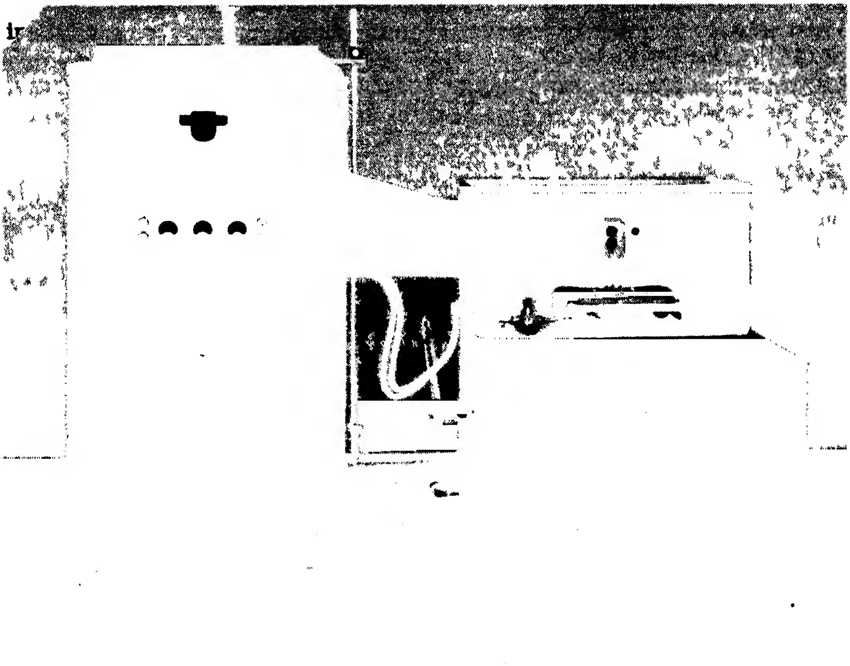


Fig 22 Typical Generator of the Vacuum Tube Type
Courtesy of the Induction Heating Corp

In (A) of Fig. 23 is illustrated one type of turning tool which may crack because of the uneven expansion and contraction of the dissimilar materials of the tip and tool shank. In this design, the shoulder, X, will cool faster on the outer surface than on the inner, thus creating a powerful working effect which will tend to curl that part of the tool point. This exerts a pulling or rending force on the carbide tip, sometimes sufficiently great as to cause it to crack along the line indicated in the figure at B. This fracture is known as a brazing crack. It can be partly eliminated by brazing a soft metal shim between the two surfaces.

In (B) of Fig. 23 is illustrated another type of carbide-tipped tool which is likely to crack either in brazing or in grinding, the crack usually developing along the line indicated at X. However, this tendency for cracking can be eliminated by sandwich brazing as was described for the turning tool.

In (C) of Fig. 23 is shown another tool which may crack in brazing and grinding, the crack usually occurring along line X. Sandwich brazing, using a soft material with a high degree of elasticity, may eliminate much of the danger.

The design shown in (D) of Fig. 23 is still another which often results

in cracked tips in brazing. The crack in this instance would be across the narrowest part of the tip. The answer to this particular problem is two-fold Either sandwich brazing or a change in the design of the tool would bring about a satisfactory solution.

Details of Sandwich Brazing. This operation consists of braz-

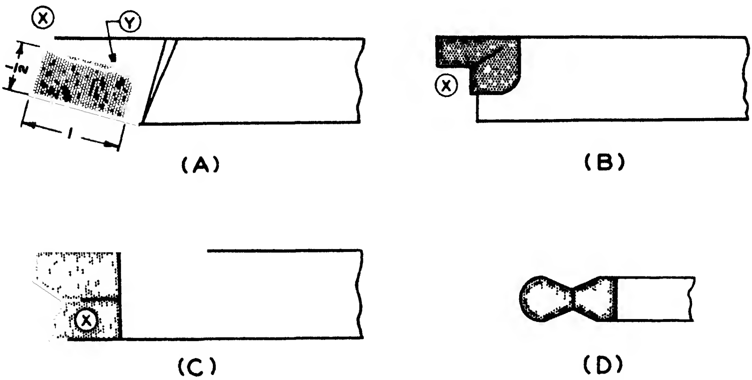


Fig. 23. Certain Types of Carbide Tipped Tools Are Especially Subject to Cracks Resulting from Brazing Stresses

ing between the tip and the shank, a shim of soft material possessing a high degree of elongation or stretching ability. The function of this shim is to absorb the strain caused by the unequal contraction of the carbide in the steel shank, thus preventing cracks. A typical sandwich braze is shown in Fig. 24. In order to be effective, the shim should be placed not only on the bottom of the tip but also on the sides, and must be brazed to both the tip and the shank.

Shim material for sandwich brazing of carbide tools usually is made of one of the following: commercially pure copper, Driver Harris Alloy #52, Constantan, stainless steel, or Monel metal. To be effective, the material must be of adequate thickness. Some tool engin rs recommend shims be only .005'' thick; others recommend sh from .015'' to .013'' in thickness. Careful study reveals that b be expected when shims of .021'' to .031'' thick are u to absorb most of the strain imposed on the tool b Commercially pure copper in the soft state is f sintered carbide tools, the other metals follo order given.

Brazing Forming Tools. Circular, d ing tools, commonly used in screw mac sandwich brazed as shown in (A) and (F material is said to absorb much of the s against the shocks encountered in the c this procedure was found to prolong

On the basis of experience, then, it is recommended that copper shim stock .031" thick be used. One large manufacturer, having hundreds of hand and automatic screw machines, reports that excellent results are obtained with his forming cutters shimmed with soft copper of this thickness.

Enclosed tip tools should be avoided by the designer or tool engineer wherever possible. Tools of this kind, shown in (B) and (E) of Fig. 26, are likely to crack either in brazing, grinding, or in use, the crack appearing along lines X - X or Y - Y as illustrated in the small drawing to the right of (B). If such a tool is the only solution to a machining problem, sandwich brazing certainly should be used.

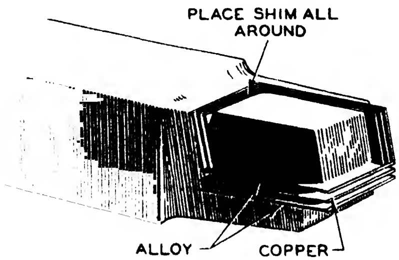


Fig. 24. A Typical Sandwich Braze

Generally speaking, the tool in (B) of Fig. 26 is inferior to the one shown at (E). Milling of the recess would be more costly and the carbide tip would probably be more subject to cracking due to the unequal expansion and contraction which would be present in a shank of this kind. A cheaper, more efficient method of doing the job is shown in (C), (D), and (E). The first step is to machine the shank flat across the top as shown at (C). The carbide tip is then put in place, (D), and steel inserts laid in the remaining space. These steel inserts are made separately and brazed to the shank and to the tip at the same time the carbide tip is brazed. This construction provides added resistance against side thrust.

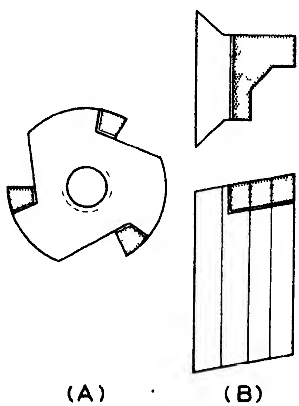


Fig. 25. Sandwiched Brazed Circular and Dovetail Forming Tools

QUESTIONS TO HELP YOU

The fo' ·· ing questions have been com·· in your reading. The
chapter have been
If you are unable
, it is suggested
al just covered.
major steps in
rbide tools?
lly formed?
igles milled to more than the specified
ced in the tool?
sides milling, of forming the recess?

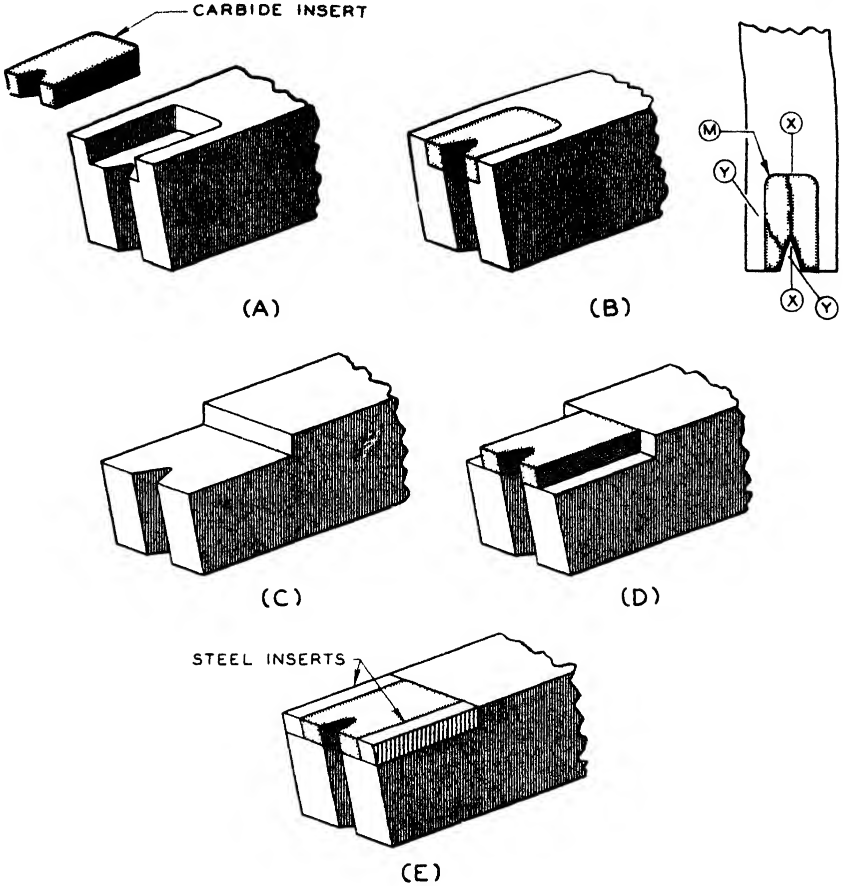


Fig. 26. Tools with Enclosed Tips Should Be Avoided

5. Does heat-treating a cold-drawn shank add to its resistance to deflection?

6. Does the use of an alloy steel, such as high-speed steel, add anything to the shank, so far as resistance to deflection is concerned?

7. What are the three common methods of brazing tips to tool shanks?

8. What is brazing?

9. What is "wiping" or "sweating" in brazing?

10. What are the common brazing materials for joining carbide tips to shanks?

11. Why is the cleanliness of the tip and shank essential?

12. Describe the torch brazing method in detail.

REFERENCES

Carboloy Company. Engineering Bulletin GT-133-R. March, 1934.

Curtis, F. W. High-Frequency Induction Heating. New York: McGraw-Hill Book Company, Inc., 1944.

Firthite Sintered Carbide Cutting Tools. Users' Handbook.

Handy and Harmon. Low Temperature Silver Brazing Alloys.

Lindenmeyer, R. S. "Brazing and Allied Processes," Science and Mechanics, August, September, October, November, 1947.

Phillippi, A. K. "Electric-Furnace Brazing," Westinghouse Engineer, Vol. 5, May, 1945.

Van Nordstrand and Others. "Factors Controlling the Strength of Brazed Joints," National Research Council Report #560.

Vascoloy-Ramet. "Tantung Tool Catalogue."

CLAMP METHOD OF AFFIXING THE CARBIDE TIP TO THE SHANK IS SOMETIMES PREFERRED TO BRAZING

Courtesy of Kennametal, Inc

CHAPTER V

Grinding Single-Point Tools

The Grinding Problem. The sintered carbides used in cutting tools are extremely hard, and, because they are hard, they are also brittle. A compensating factor is their great compressive strength but their heat conductivity is only about half that of steel. Because of their low tensile strength and lack of ductility, the carbides tend to crumble at the cutting edge. They also "check" during grinding. Their low heat conductivity makes it important that localized heating be avoided both in brazing and grinding. As a result, special grinding methods and careful selection of grinding wheels are necessary. This chapter will explain all these matters in detail.

The problems in grinding tools tipped with sintered carbides are different from those involved in grinding carbon-steel or high-speed-steel tools. The problem concerning any tool is the economical production of a fine cutting edge which is properly supported and is free from surface cracks and checks. Since the carbides differ radically from high-speed steel, they require different grinding wheels and methods.

In the first place, the material is much harder than high-speed steel, being 74 to 82 on the Rockwell C scale of hardness as compared with 62 to 65 for high-speed steel. The variation in hardness of the carbide depends on the grade, there being harder and softer grades for various uses. Because of its greater hardness, then, the material is more difficult to grind. More heat is generated in grinding carbide tools than in grinding high-speed-steel tools. Hence, there is greater chance for overheating and consequent cracking of the tip.

Grinding Wheels Used. For grinding sintered carbides, hard and sharp abrasives are needed, and wheels are used that are quite different from those in common use in tool rooms. These wheels are made either of friable silicon carbide or of diamond particles. In addition, the wheels are so made that their structure is open and porous. This is particularly true in the case of the silicon carbide wheels. The combination of sharp and friable abrasive with open bonding insures a wheel with remarkably cool cutting qualities which results in economical sharpening of tools. When an especially cool wheel action is required, diamond wheels with resinoid, vitrified, or metal bond are used to advantage.

Occasionally it becomes necessary to grind the steel shanks support-

ing the carbide tip. Aluminum oxide wheels are used for this job. This type of wheel should have either a resinoid or a vitrified bond. The aluminum oxide wheel provides a faster and more economical way of grinding the tool shank. Silicon carbide and diamond wheels will grind the steel supporting shank but they soon become loaded with steel dust, requiring redressing of the silicon carbide wheel and cleaning of the diamond wheel. Not a few diamond wheels have been ruined by this practice.

Characteristics of the Wheels. All grinding wheels have two distinct characteristics. These are the grain or grit size (fineness of the abrasive particles), and the grade or hardness of the bond.

By grain or grit size is meant the size of the cutting particles. This size is represented by a number which denotes the mesh of the screen used in preparing the abrasive. The size of the cutting particles in

TABLE I. STANDARD ABRASIVE GRAIN SIZES

Coarse	Medium	Fine	Very Fine
10	30	70	220
12	36	80	240
14	46	90	280
16	54	100	320
20	60	120	400
24		150	500
		180	600

silicon carbide wheels determines the wheel's degree of fineness or coarseness. This, in turn, is determined by the use to which the wheel is put—whether it is used on a roughing or finishing operation. A combination of grain sizes is often used in a wheel in order to get the best results for a particular job.

The standard abrasive grain sizes are determined by the openings per linear inch in the sieves through which the abrasive particles are screened. The grain size conforms to U.S. Department of Commerce specifications. The standard grain sizes are grouped into coarse, medium, fine, and very fine classifications. This grouping is shown in Table I.

Grade, on the other hand, is the measure of the strength of the bond or the resistance it offers to the force which tends to break it down while in use. In the manufacture of the wheel, the bond is mixed with the abrasive. The material is then shaped to form in a hydraulic press, after which the wheels are heated. This heating hardens the bond. The bonding material forms a complete coating over each abrasive particle, causing it to adhere more or less tenaciously to the adjacent particles, The thicker the film of bonding material, the stronger will be the bond, and, therefore, the harder the wheel. The amount of bond is indicated

by the letter symbol in the marking on the wheel. The range of bond grades is indicated in Table II. It should be pointed out that the most commonly used wheels generally range from soft to medium when carbide is being ground, and from medium to hard when high-speed steel is being ground.

Standard Marking System. In December of 1944, the manufacturers of grinding wheels adopted a new system for marking grinding wheels, devised after much study by the standardization committee of

TABLE II. GRADES OF GRINDING WHEELS

Very Soft	Soft	Medium	Hard	Very Hard
A	F	K	P	U
B	G	L	Q	V
C	H	M	R	W
D	I	N	S	Y
E	J	O	T	Z

the Grinding Wheel Manufacturers' Association. Table III graphically illustrates the new standards system adopted by them. This system does away with much of the confusion which existed previously as the result of the different methods used by the various manufacturers in marking their products.

The marking system comprises six positions in addition to the manufacturer's prefix, which indicates his formula for the abrasive. In the first position after the prefix, the letter symbol represents the abrasive used. For example, A is aluminum oxide, regular; B is aluminum oxide, refined; AB is a mixture of regular and refined aluminum oxide; C is silicon carbide, regular; CC is silicon carbide, refined; D is corundum; and E is emery.

The designation in Table III is for an alundum (aluminum oxide) wheel, indicated by the letter A in the first position. The grit size is given in the second position, indicating a grain size of 36. The grade or the hardness of the wheel is given in the third position, the letter L indicating a wheel of medium hardness. The fourth position denotes the structure of the wheel, 1 being the most dense and 15 the most open. The fifth position gives the type of bond used in accordance with the list in the table. The sixth position is reserved for the manufacturer's record number and is considered optional.

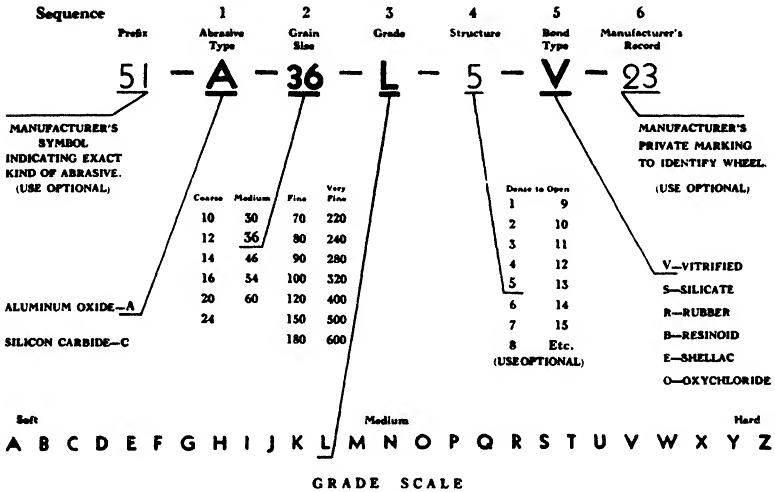
Assuming a grinding wheel is marked 50 - C - 60 - H2 - B - 20, the characteristics of this wheel, according to the information just presented, would be as follows:

50 is the manufacturer's prefix which indicates his formula for the abrasive used in the wheel

C denotes that the abrasive is silicon carbide, regular

60 is the grain size of the abrasive particles

TABLE III. STANDARD MARKING SYSTEM CHART



Courtesy of the Norton Company

H is the hardness of the bond (medium soft)
2 is the density of the structure (extra dense)
B indicates the wheel to be made with a resinoid bond
20 is the manufacturer's record number

It should be observed that items such as the type of abrasive used, the grain size, the grade, structure, and the type of bond are clearly marked, since these are the most important points in wheel identification. These characteristics assume even greater significance in the grinding of sintered carbide tools.

Silicon Carbide Wheels. Silicon carbide wheels are commonly used for rough as well as finish grinding of sintered carbide tools. For ordinary offhand grinding and for machine grinding, an open, porous structure is needed. A wheel which is too hard may result in localized heating. This may cause the tool tip to check and crack. This condition may be avoided through the use of a fairly soft wheel. However, the wheel should not be so soft that it will wear excessively. Thus, it is necessary to choose a wheel that is a compromise between one which will dull and heat the work, and one that will wear too rapidly. Because of their lower first cost, silicon carbide wheels of the proper grain size are frequently preferred over the diamond wheel.

The most commonly used wheels for grinding single-pointed tools are the straight and the cup types, shown in (A) and (B) of Fig. 1. The straight wheel, used for the roughing of work in offhand grinding and for finishing in machine grinding, possesses one important advantage over the cup-type wheel. It has less surface contact with the work, and, therefore, has a somewhat cooler cutting action.

In (A) and (B) of Fig. 1 are shown the most commonly used types of wheels for grinding carbide tools. The key to the letter dimensions shown is as follows:

A is the flat spot of beveled wall
D is the diameter
E is the thickness of hole
H is the hole
T is the thickness over all
V is the angle of bevel
W is the wall thickness at grinding face

When ordering a grinding wheel, it is necessary to give the specification, the size, and the type desired. Thus, for 10 straight wheels for rough grinding carbide tools, the order would be placed as follows:

Ten grinding wheels, type 1, C - 60 - J6 - V, size 7'' × 1/2'' × 1 1/4''.

This indicates that the wheel is silicon carbide, grain size 60, J grade, density 6, and the bond is vitrified. The diameter is 7'', thickness 1/2'', and the hole is 1 1/4'' diameter.

The code GC - 80 - J6 - VW indicates by the letter G the kind of silicon carbide used, and by letter W, the manufacturer's identification of the wheels. Once these have proven successful on the job, and are reordered, all this information should be placed on the order.

The standard grain sizes generally used in silicon carbide wheels for grinding the carbides are 60, 80, 100, 120, 150, and 180. The coarser the grain size, the higher is the rate of stock removal, but the rougher is the finish obtained. Conversely, the finer the grain size, the slower is the rate of stock removal, but the smoother is the finish obtained. The finer wheels are more likely to overheat the carbide tip and crack it.

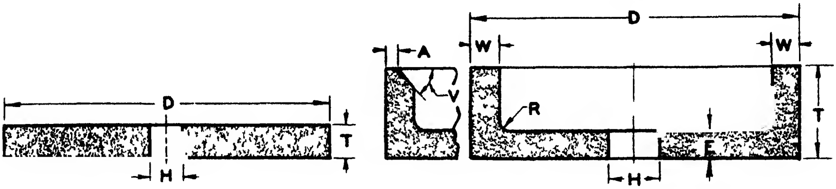
Diamond Wheels. Diamond wheels in resinoid, metal, and vitrified bond are being used increasingly for sharpening carbide tools. They possess certain advantages, among which are the more rapid cutting with slower wear than is possible with silicon carbide wheels; the elimination of some cracking and checking because of the sharp, free cutting action; possible elimination of the finish grinding operation because the finish obtained with diamond wheels is usually adequate; and avoidance of the overheating of the carbide tip during grinding, thereby lessening the chances of thermal or heat cracks and checks.

The finishes obtained with diamond wheels depend on the grain size used and the feed of the work. Standard grain sizes of diamond wheels commonly used in grinding the carbides are 80, 90, 100, 120, 150, 180, 220, 320, 400, and 500. The 80 to 180 grain wheels are used for grinding tools requiring ordinary finish on the cutting edge. The 220 and 320 grain wheels are used for finish grinding tool edges which were previously roughed out on other wheels. The 400 to 500 grain wheels are used for lapping or for giving a very fine finish to the cutting edges of tools.

Diamond Concentration. Diamond wheels have another designation, that of diamond concentration. Diamond concentration is the

quantity of diamonds present in a unit volume in the diamond section of the wheel. The quantity in a unit section is designated by the numbers 25, 50, and 100. Fig. 2 illustrates the relative concentration of diamonds in the three classifications.

The diamond wheel is so constructed that the depth of the diamond



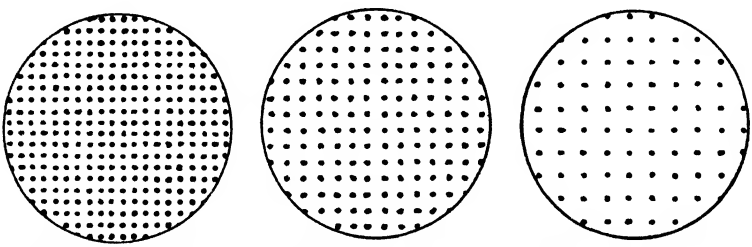
(A) (B)

Fig. 1. The Straight Grinding Wheel (A) and the Cup Wheel (B)

section of the wheel is from 1/32" to 1/4". The reason for this is one of economy. Obviously, a greater depth of the diamond section in any concentration results in an increased amount of diamond grains in the wheel. This, of course, increases the cost of the wheel. The construction of the diamond wheel is shown in (A) and (B) of Fig. 3.

Diamond Wheel Marking System. Unfortunately, the systems for marking diamond wheels are not as well standardized as those used in marking the silicon carbide wheels. Table IV is the diamond wheel marking system devised by the Norton Company. There are several others in use by different manufacturers.

Choice and Use of Grinding Wheels. Recommendations are made in Table V governing the use of silicon carbide and diamond wheels in sharpening sintered carbide, single-point tools both by off-



"100" Concentration *"50" Concentration* *"25" Concentration*

Fig. 2. Relative Concentration of Diamonds in Grinding Wheels

Courtesy of the Norton Company

hand and machine methods. Because of the varying conditions that are peculiar to each job, such as the skill of the operator, the condition of the machine used, and whether the grinding is done wet or dry, the recommendations in Table V are only approximate. However, these

recommendations do provide a good starting point in the selection of wheels, and it will be found that the suggestions are very close to the needs of the job indicated. Experimentation will determine the wheel best suited for a special job. The various manufacturers of grinding wheels prepare booklets which give close specifications for their prod-

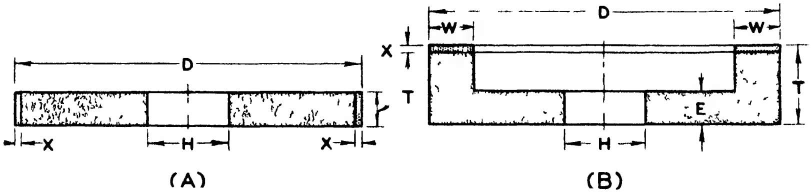


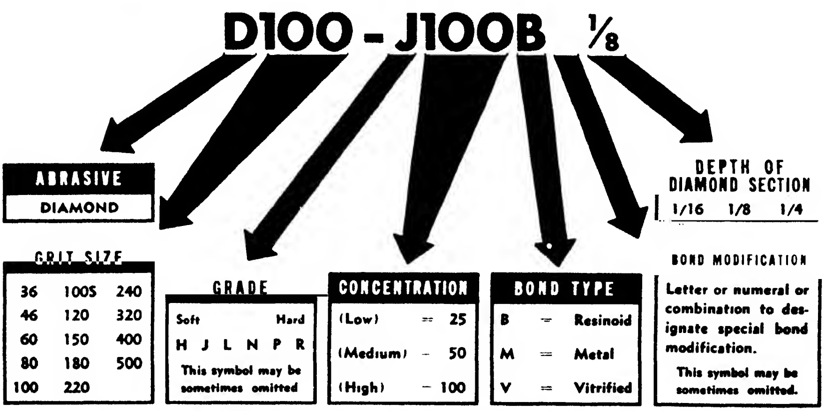
Fig. 3. Construction of a Straight Diamond Wheel (A) and the Cup-Type Wheel (B)

ucts. Wherever possible, the information contained in these booklets should be followed.

Sintered carbide tools have a tendency to chip easily at the cutting edge, or, in fact, at any sharp edge. It should again be emphasized that this condition can be obviated through the use of a soft wheel in all grinding operations. When the contact between the wheel and the carbide is slight, a somewhat harder wheel may be employed. Coarser wheels are used for heavy stock removal. When a finer finish is required, wheels should be used that are made of finer grit or grain sizes. In all cases, the wheels should never be too hard or too fine in grain size

TABLE IV. NORTON DIAMOND WHEEL MARKING

Analysis of Typical Norton Diamond Wheel Specification



HAND HONES will be marked with grit size, bond type and depth of diamond section (No grades or concentration will be shown)

Courtesy of the Norton Company

since the use of such wheels tends to overheat the tool. On the other hand, too soft a wheel should not be selected as it will wear off too rapidly and may not leave a satisfactory edge.

TABLE V. WHEELS RECOMMENDED FOR GRINDING CARBIDES

Type of Operation	Abrasive Used	Grain Size	Grade	Porosity	Bond
Offhand					
Roughing	SiC	60	H,I,J	6	vitrified
	diamond	100	J		resinoid
Finishing...........	SiC	100	H,I,J	5	vitrified
	diamond	100	J		resinoid
Machine Grinding					
Roughing...........	SiC	60	G,H,I	5-6	vitrified
Finishing...........	SiC	80-100	H,I,J	5-6	vitrified
Surface Grinding					
Roughing, dry	SiC	60	G,H,I	6	vitrified
Finishing, dry....	SiC	100	G,H	5	vitrified
Roughing, wet	diamond	100	J		resinoid
Finishing, wet....	diamond	220	J		resinoid
Form Grinding, wet...........	SiC	100-120	J,K	5	vitrified
Lapping					
Average lap.......	diamond	220-320	H,I		resinoid
Fine lap............	diamond	400-500	H,I		resinoid
Extra fine	nickel chromium cast-iron disc charged with #4 diamond dust				

Grinding Wheel Speeds. By grinding wheel speeds is meant the speed of the surface of the wheel in contact with the work. This speed can be calculated from the following formulas:

$$V = \frac{3.14 \times D \times N}{12}$$

$$N = \frac{V \times 12}{3.14 \times D}$$

In which

V = the peripheral speed of the wheel in f.p.m.
D = the diameter of the grinding wheel in inches
N = the revolutions per minute (r.p.m.) of the wheel

To illustrate the use of these formulas, assume it is desired, first, to find the speed of a 6'' wheel which is revolving against the work at 3,000 r.p.m. The known factors are D = 6'' and N = 3,000. V is the value to be solved for. Applying the first formula and substituting the known factors for the letters gives

$$V = \frac{3.14 \times 6 \times 3{,}000}{12} = 4{,}710 \text{ f.p.m.}$$

Again, assume it is desired to find at what speed an 8'' wheel should revolve in order that its peripheral speed be 4,000 f.p.m. The known factors are V = 4,000 and D = 8''. N is the value to be solved for. Applying the second formula and substituting the known factors for the letters gives

$$N = \frac{4{,}000 \times 12}{3.14 \times 8} = 1{,}910 \text{ r.p.m.}$$

In general, grinding wheel speeds for sharpening the single-point, sintered carbide tools are between 4,000 and 5,500 f.p.m. Certain machines with special spindles operate satisfactorily at wheel speeds from 2,500 to 4,000 f.p.m. Table VI gives r.p.m. for grinding wheels from 1'' to 42'' in diameter and speeds from 4,000 to 7,500 f.p.m.

TABLE VI. R.P.M. AND PERIPHERAL SPEED IN F.P.M. FOR GRINDING WHEELS OF VARIOUS DIAMETERS

Wheel Diam., Inches	Peripheral Speed in F.P.M.							
	4,000	4,500	5,000	5,500	6,000	6,500	7,000	7,500
1	15,279	17,189	19,098	21,008	22,118	24,828	26,737	28,647
2	7,639	8,594	9,549	10,504	11,459	12,414	13,368	14,328
3	5,093	5,729	6,366	7,003	7,639	8,276	8,913	9,549
4	3,820	4,297	4,775	5,252	5,729	6,207	6,685	7,162
5	3,056	3,438	3,820	4,202	4,584	4,966	5,348	5,730
6	2,546	2,865	3,183	3,501	3,820	4,138	4,456	4,775
7	2,183	2,455	2,728	3,001	3,274	3,547	3,820	4,092
8	1,910	2,148	2,387	2,626	2,865	3,103	3,342	3,580
10	1,528	1,719	1,910	2,101	2,292	2,483	2,674	2,865
12	1,273	1,432	1,591	1,751	1,910	2,069	2,228	2,386
14	1,091	1,228	1,364	1,500	1,637	1,773	1,910	2,046
16	955	1,074	1,194	1,313	1,432	1,552	1,672	1,791
18	849	955	1,061	1,167	1,273	1,379	1,485	1,591
20	764	859	955	1,050	1,146	1,241	1,387	1,432
22	694	781	868	955	1,042	1,128	1,215	1,302
24	637	716	796	875	955	1,034	1,115	1,194
26	588	661	734	808	881	955	1,038	1,101
28	546	614	682	750	818	887	955	1,023
30	509	573	637	700	764	828	891	955
32	477	537	597	656	716	776	836	895
34	449	505	562	618	674	730	786	843
36	424	477	530	583	637	690	742	795
38	402	452	503	553	603	653	704	754
40	382	430	477	525	573	621	668	716
42	364	409	455	500	546	591	637	682

The safe speed for a grinding wheel is usually stamped on the blotter washer, which normally is glued to the wheel. This speed should not be exceeded because of the possibility that the wheel might burst. Bursting of the wheel would not only damage the work but would also endanger the safety of the operator.

Mounting and Truing the Wheel. Straight silicon carbide wheels should be mounted on the spindle so that they will run as nearly true as possible. Once they are mounted they are usually trued carefully with a diamond tool while the wheel is revolving on its own spindle. This operation is shown in Fig. 4. When this has been done, the wheel is ready for cutting.

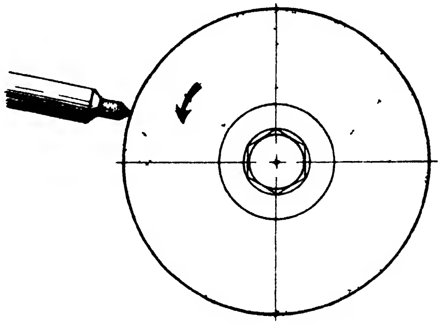


Fig 4. Truing a Silicon Carbide Wheel with a Diamond Dresser

Diamond wheels, on the other hand, cannot be corrected by truing with a diamond tool. For this reason, they must be mounted on the spindle so that they run true to within .0005''. This is done as follows: The arbor screw holes in the straight diamond wheels are a few thousandths of an inch oversize to permit shifting of the mounting so the periphery can be made to run true. The wheel

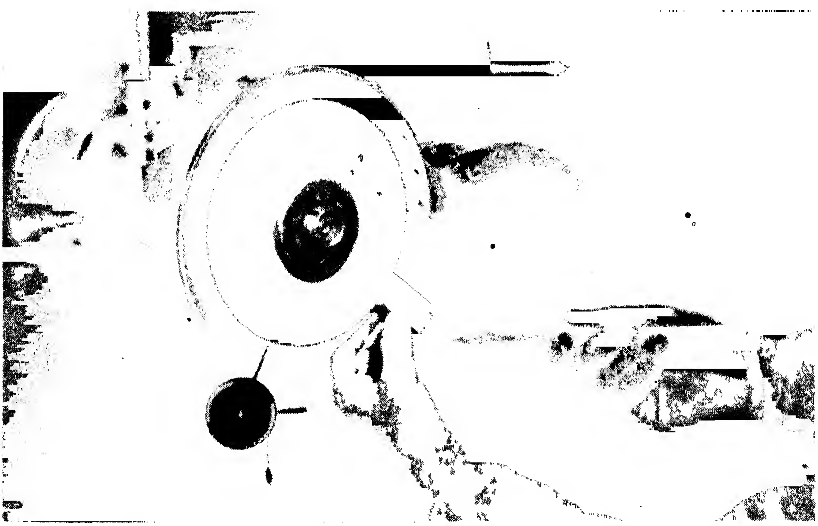


Fig. 5. Method of Truing a Diamond Wheel on Its Spindle

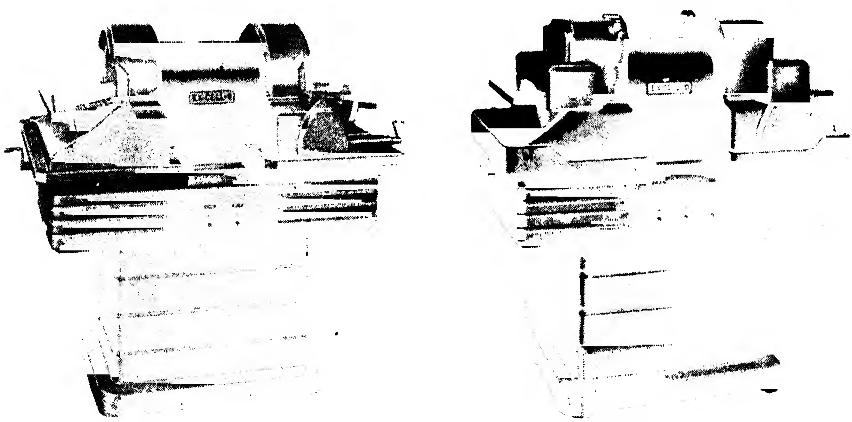
is first placed on the spindle and tightened lightly. The periphery is then tested with a dial indicator as shown in Fig. 5. If the wheel runs true, it is tightened securely and used. If the wheel does not run true, the high spot is tapped with a light hammer or mallet, a block of wood

Fig. 6. Dressing Stick for Diamond Wheels

being used to cushion the blow. This, also, is shown in Fig. 5. The wheel is again checked with the dial indicator. This process is repeated until the wheel runs true to within .0015". Once this is achieved, the flange is tightened securely against the wheel web and the wheel is ready for truing with diamond as shown in Fig. 4.

Diamond wheels, properly set, seldom need dressing. When they become glazed or loaded, they sometimes may be restored to their original sharpness by brushing the surface with kerosene or by dressing it lightly with a stick of pumice. When it becomes necessary to change the shape of the wheel face for form grinding, it should be ground with a silicon carbide wheel of about 46 to 60 grit size or dressed with a dressing stick, one type of which is shown in Fig. 6.

To restore the sharpness of the cup-type diamond wheel, the rim of the wheel can be lapped with 180 grain size silicon carbide abrasive which is mixed with water and is spread on a cast-iron surface plate. This will also help restore the plane or trueness of the surfaces.



(A) (B)

Fig. 7. A Heavy-Duty Grinder with 14" Wheels (A) and a Similar Grinder (B) That May Be Used for Either Wet or Dry Grinding

Courtesy of Ex Cell O Corporation

Feeds. For surface grinding with straight wheels, the down, or vertical, feed should be from .00025'' to .001'' per pass. The table traverse should be from 100'' to 500'' per minute, and the cross feed from .036'' to .060'' per pass. The vertical feed and the cross feed should be reduced for finishing cuts in order to get a better finish on the tool surface. Excessive vertical feed may cause overheating which can easily crack the carbide tip. When the vertical feed is .001'', a large amount of coolant should be used to prevent the tool from being injured.

Coolants. Resinoid wheels will char at temperatures above 600° F., and a coolant should be used to keep them in condition. This coolant may be plain water, ordinary grinding solution, or kerosene. When wet grinding equipment is not available, the wheel may be lubricated with a felt wick which is saturated with oil and held in contact with the wheel by a spring. Where the contact between the wheel and the work is slight, as in the case of small, straight, grinding wheels, dry grinding may be done without danger of overheating and cracking the tip.

When grinding wet with vitrified silicon carbide wheels, sufficient fluid should be used to keep the work and the contacting part of the wheel well flooded and cool. Inadequate or intermittent cooling causes strains and cracks. Therefore, sharpening of tools by grinding should be done either all wet or all dry. It is necessary when grinding dry to exercise great care not to overheat the carbide tip as this will invariably cause it to crack or check.

Grinding Machines. Grinding machines used for the sharpening of single-point tools may be classed in two general groups: machines for offhand grinding, and surface grinders used for mechanical grinding. In (A) of Fig. 7 is shown a typical machine which is used for dry, offhand grinding. In (B) is pictured a machine which is used for wet, offhand grinding.

It is desirable, in offhand grinding, that the machine be rigid and free from vibration. A loose spindle may easily result in inaccurate work or in checked or cracked tool tips because of the vibratory action of the grinding wheel on the tool tip.

In Fig. 8 is illustrated another type of machine which is used in the offhand sharpening of carbide tools. This machine is equipped with adjustable, graduated tables or tool rests. Such tables have keyways or grooves in their surfaces for guiding a fixture in the event one is used for holding the tool. These fixtures support the tool and insure that the angles to which the tools are to be ground are accurate.

A number of the surface grinding machines are suitable for sharpening carbide tools accurately and economically. In Fig. 9 is illustrated a Covel surface grinding machine which is suitable for hand operation. Fig. 10 shows an automatic feed and traverse machine of the same make. There are many other good ones. In either the hand or the automatic feed machine, the tool is held for grinding in a fixture, a vise, or on a magnetic chuck.

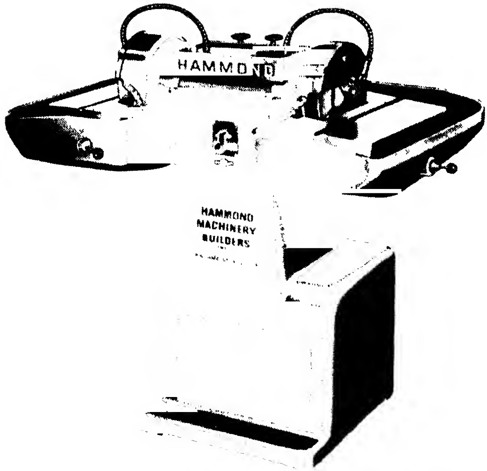


Fig. 8. This Grinder May Be Used for the Offhand Sharpening of Tools

Courtesy of Hammond Machinery Builders, Inc

Fig. 9. Hand-Feed Surface Grinding Machine Which Can Be Used for Sharpening Carbide Tools

Courtesy of the Covel Manufacturing Company

Offhand Grinding. Offhand grinding may be used both for roughing and for semifinishing. When the quantity of tools to be ground is not large, either the periphery of a straight wheel or the face of a straight cup wheel may be used for roughing. When the periphery of a straight wheel is used, it is dressed to a crown as in Fig. 11. This will reduce the surface contact of the tool on the wheel.

Carbide tools are usually designed and constructed so that no grinding of the steel shank is necessary. There is usually provision for a slight overlap of the carbide tip. This permits the grinding of the tip only, saving time and wheel wear. The overlap is shown clearly in Fig. 12. The amount of overlap on single-point tools should not be greater than 1/64" after the tool is finished. This overlap should be less for those tools with small tips.


Fig. 10. Covel Automatic Surface Grinding Machine
Courtesy of the Covel Manufacturing Company

When a tool has become badly chipped as the result of the necessity for removing a large amount of stock, the rough grinding in repointing the tool can be done on a straight wheel as shown in Fig. 13. The carbide portion of the tool is ground first, from top down, after which the shank part of the tool is ground. This is shown in Fig. 14. When enough material has been removed, the front of the tool will resemble an open or wide V shape when viewed from the side. This is ground

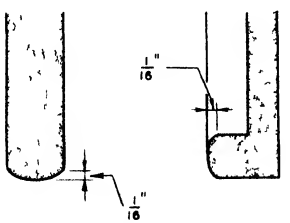


Fig. 11 Crowning the Roughing Wheel

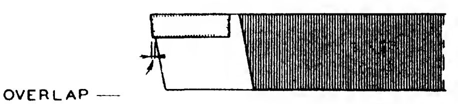


Fig. 12. Design of Tool for Grinding Tip Only

off by setting the tool flush to the wheel in a way similar to that shown in Fig. 14. This step in the grinding will produce a tool which is hollow shaped on the end as shown in Fig. 15. The amount of concavity will

depend on the diameter of the grinding wheel used. This concavity can be easily removed in the semifinishing operation.

Semifinishing of Tools. After the roughing out has been done, the tool may be semifinished on a cup wheel if it is found to be necessary. This is done by first grinding the top of the tool, then the front, and finally the side, the table being set at the desired relief or rake angle. The pressure of the tool against the wheel should be light in this

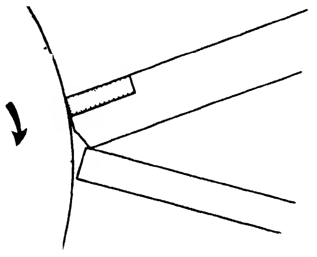


Fig. 13. Rough Grinding the Tip on a Straight Wheel

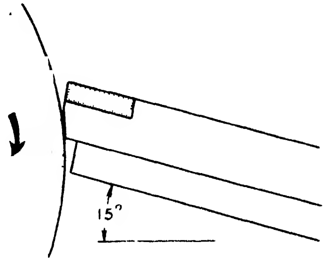


Fig. 14. Rough Grinding the Shank of the Tool

operation because the contact surface between the tool and the wheel is great and there is danger, therefore, of overheating and cracking the tip. Fig. 15 shows the roughed-out tool, while the sequence of the semifinishing operations is illustrated in (A), (B), and (C) of Fig. 16.

In all carbide sharpening, the grinding wheel should rotate so that cutting is done from the cutting edge toward the body of the carbide. This important point is illustrated by the wheel rotation direction arrows in Figs. 13, 14, and 16. If this procedure is not followed, the edge of the carbide tip may be chipped.

A semifinished tool is entirely satisfactory for many cutting operations. When a better finish is required, the tools are finish-ground on a 180-grit silicon carbide wheel, or on a 220-grit diamond cup wheel.

Grinding Procedure. The step-by-step process for roughing when using a 60-grit silicon carbide wheel is as follows: First, the face of the tool is ground. Following this, the front relief angle is rough-ground to about 4° more than the finished relief angle, leaving a land of about 1/32" for finishing. The side relief angle is then rough-ground to about 10°, and a 1/32" land left on the side for finishing.

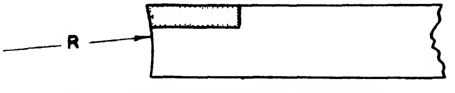


Fig. 15. The End of a Roughed-Out Tool

The step-by-step procedure for semifinishing when using a 100-to-120-grit silicon carbide wheel or 100-grit diamond wheel is as follows: First, the table is set to the desired rake angle and the top of the tool is finished to the required angle. Following this, the front relief is ground.

The direction of wheel rotation should be checked to make certain that it revolves against the cutting edge of the carbide tip. This will obviate any possibility of chipping the edge of the carbide tip. The table is then set for the side relief angle and the tool ground accordingly, again making certain that the wheel rotation is against the cutting edge. If the tool

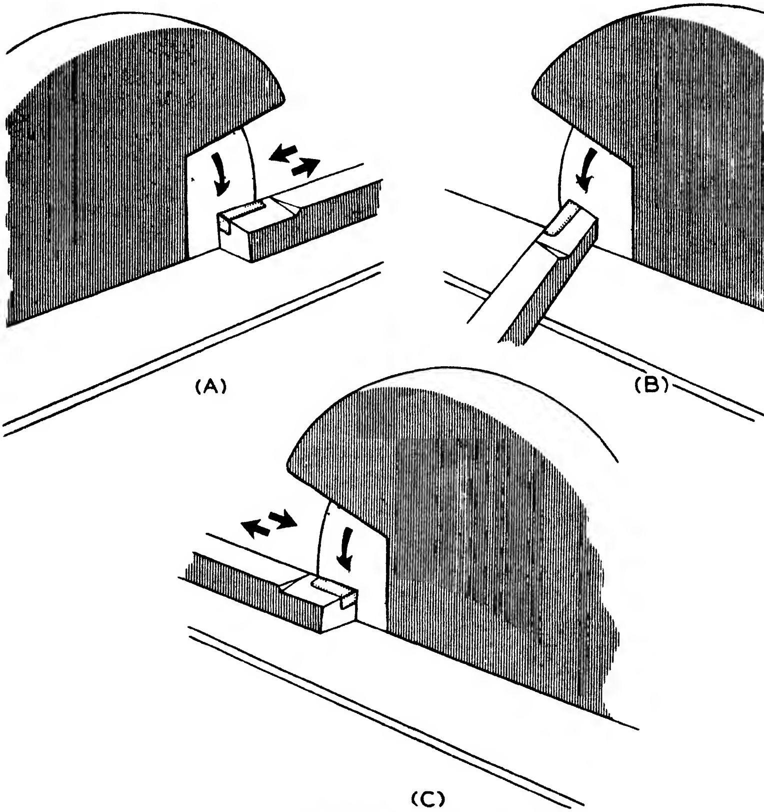


Fig. 16. The Sequence of Semifinishing Operations

is to be used in the semifinished condition, the nose radius should now be ground. This is done by swinging the tool body in an arc on the table as shown in Fig. 17, holding the tool point against the wheel.

Tool Lapping. When it becomes necessary to have the cutting edge ground to a high degree of finish, grinding with a 220-grit diamond wheel or a 180-grit silicon wheel will be necessary. This is known as finish grinding, sometimes called lapping. The function of finish grinding or lapping is to remove any unevenness on the tool surfaces and to present a better cutting edge against the work. It is a well-known fact

that the smoother the cutting edges and surfaces, other conditions remaining the same, the better the finish obtained by the tool, and, what is even more important, the longer is the tool life.

In fine, grind-finish operations or lapping, the procedure and the sequence are the same as detailed for semifinishing. That is, the grinding is done first on the top, then on the front, and lastly on the side. It is essential that the cutting always be done from the edge of the tool.

Machine Grinding. When large quantities of tools are to be produced, or when dimensional accuracy is to be maintained, offhand grinding will not be found to be either satisfactory or economical. Again, for grinding tools having forms such as those shown in Figs. 18 and 29, offhand grinding will not be found to be satisfactory, for there is danger of changing their shape unless the work is done by a highly skilled operator. Forming tools are designed to machine irregular or curved shapes otherwise difficult to machine. These tools may be relatively simple as a flat tool shown in Fig. 19, or in (A) of Fig. 29, or they may be complex as that shown in Fig. 28. The flat tools of the type shown in Fig. 19 and in (A) of Fig. 29 are sharpened by grinding on the top, while tools of the tongue and dovetail type shown in (B) of Fig. 29 are sharpened by grinding at the end only, as grinding at the top would change their ability to reproduce the required shape. Again, the dovetail tool shown in Fig. 28 can be sharpened by grinding on the top only, as grinding in front would change its contour.

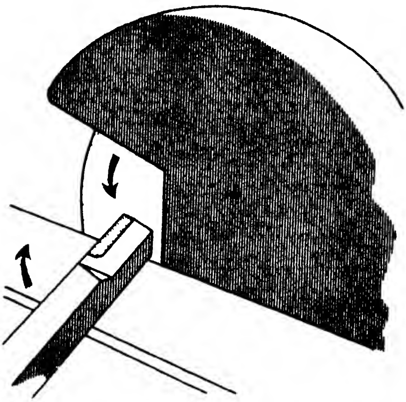


Fig. 17. Proper Direction to Grind the Carbide Tip When Forming the Nose Radius

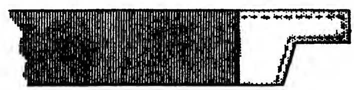


Fig. 18. A Flat Forming Tool

Since the accurate grinding of these tools is of utmost importance, it will be necessary to hold the work in a universal vise or a special vise fixture, grinding the tool in a surface grinder using a straight-type silicon carbide wheel. A vise fixture of a type in general use is shown in Fig. 20. Its application in machine grinding is shown in Fig. 19. Fig. 21 shows one of the many types of surface grinding machines which are suitable for sharpening single-point cutting tools.

All precautions previously given for hand or offhand grinding must be observed in the surface grinding of sintered carbide tools. It is important that cutting be done from the edge toward the body of the tool

tip, that the down feed of the wheel be no more than .001" per pass, and that the table traverse be as rapid as possible. Surface grinding may be

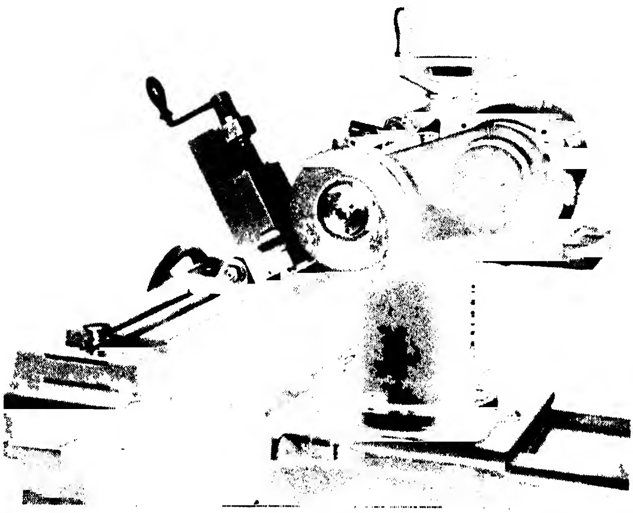


Fig. 19. Setup of a Vise Fixture for Holding Tools in Grinding *(Left)*

Courtesy of Brown & Sharpe Mfg Co

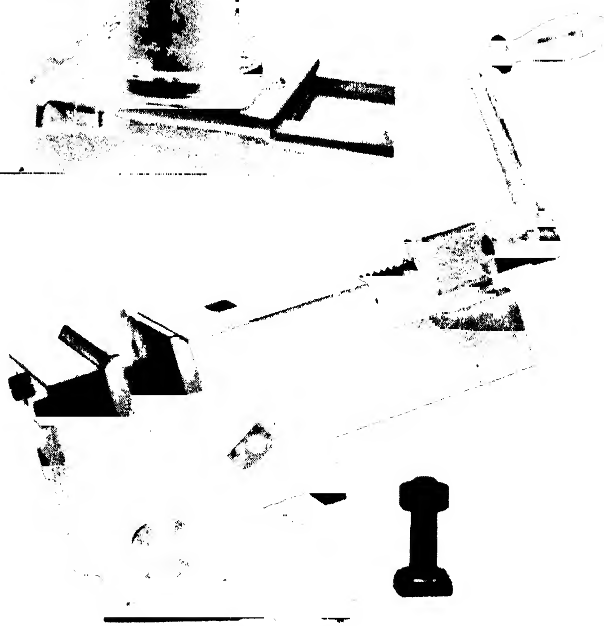


Fig. 20. Vise-Type Tool Holding Fixture Used in Grinding Forming Tools *(Right)*

Courtesy of the Norton Company

done either wet or dry. Wet grinding is to be preferred because it permits the use of harder grinding wheels with less danger of overheating and consequent cracking of the tip.

The amount of stock to be allowed for removal in the grinding of the tool tip usually is about .010" on the thickness, .015" on the width, and from .015" to .020" on the length.

Although most modern surface grinders can be adapted to wet grinding, there are quite a few machines which are specially built for this type of grinding. A special wet grinding machine is illustrated in Fig.

22. Note the two arm holding fixtures which are designed so as to exert a given amount of pressure on the tool against the wheel.

To grind the front relief or clearance angle and the front face of the tool on a surface grinder, the tool must be held in an angle-plate fixture or in a universal vise. Fig. 23 illustrates a fixture which is capable of holding tools of various sizes. It is machined to an angle of 90°– the desired relief angle and must be rigid, easily accessible, and should have convenient clamping arrangements.

Chip Breakers and Curlers. When machining steel at high speeds with single-point carbide tools, continuous chips are produced at

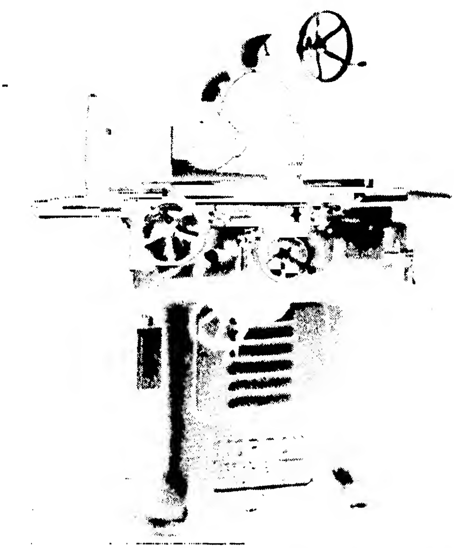


Fig. 21. Surface Grinding Machine Made by the Norton Company

Courtesy of the Norton Company

a very fast rate. These chips are dangerous to the operator and are difficult to handle. If they are allowed to go in any direction, they may wind around the revolving work, whipping against the cutting edge, chipping it. These long, continuous chips may be controlled by coiling them into springlike curls, by breaking them against the tool or the work itself, or by means of a mechanical breaker above the tool.

Coiling of the chip is done by a chip curler, commonly known as a chip breaker, which is ground into the surface of the tool as shown in (A), (B), and (C) of Fig. 24. The parallel chip breaker at (A) is used when the cut is made against the shoulder of the work. As the chip leaves the work, it is coiled against the shoulder of the chip breaker and is either forced back against the work and broken, or is led away by the

true rake angle. On shallower cuts, the chip is curled and broken against the side flank of the tool. The chip curler shown in (B) is used extensively in turning and boring operations. The function of this type of chip curler is to turn the chip against the work and break it. For light, fast, finishing cuts, a chip breaker, ground at an angle of 45°, is most frequently used. This tool is shown in (C).

Although there are other types of chip breakers, the three types shown in these sketches can be adapted to a large variety of work by

Fig. 22 Special Wet Grinding Machine
Courtesy of Hotpoint Inc

changing either the angle, the width, or the depth of the groove. Chip breakers are unnecessary when machining nonferrous materials and cast iron and should never be ground into the tool when such substances are being machined

Chip Breaker Depth and Width. The depth of the chip breaker depends largely on the feed. Normally, it should be about two-thirds of the feed and never less than 1/64''. It should be sufficient in depth to cause the chip to deflect and curl in the desired direction. Should the depth of the chip breaker be insufficient, the chip will slide over the top and the breaker will be of no use.

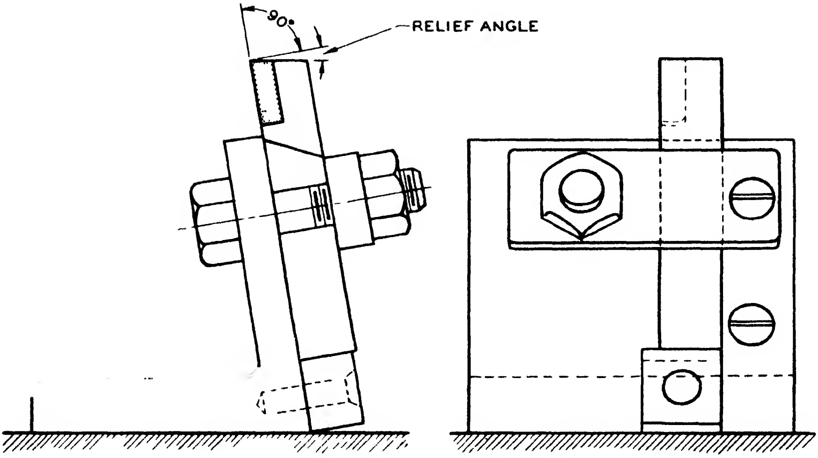


Fig. 23. Angle Plate Fixture for Grinding Front Face and Relief

The width of the chip breaker determines the diameter of the curled chip. The narrow-grooved chip breaker curls the chip tightly, while the wide chip breaker coils the chips in a wide diameter helix. The width of chip breakers for the parallel and the angular types is about one-half the depth of the cut. For the type shown in (C) of Fig. 24, the width depends on the application of the tool, but it is seldom less than 3/32".

The chip breaker should, in all cases, be wider at the point than the

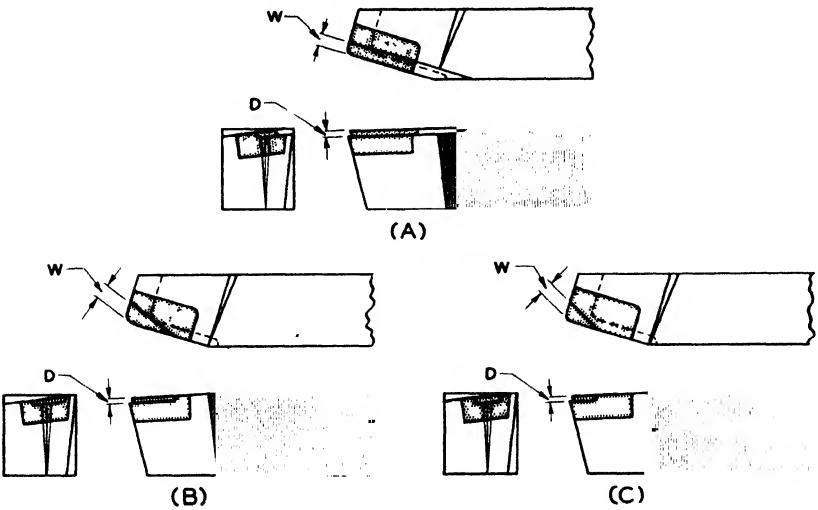


Fig. 24. Various Types of Chip Breakers

tool nose radius. This is illustrated in the tool shown in (A) of Fig. 25. If the tool nose radius is too great or the chip breaker too narrow, the point of intersection of the shoulder and the end cutting edge will contact the work before the nose radius contacts it, a situation that is not desirable. A tool having a nose radius greater than the chip breaker width is shown in (B) of Fig. 25.

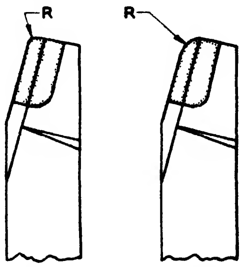


Fig. 25. Chip Breaker Width

The shoulder at the base of the chip breaker should have an adequate radius or fillet—not less than .010". Otherwise, the chip will exert pressure against the shoulder which will either chip the tool edge or flake off the top of the tool tip. Table VII gives the proper widths of chip breakers for various feeds and depths of cuts.

Grinding the Chip Breaker. In order to control the conditions under which chips curl or break, it is necessary that the chip breaker be ground each time to the same angle and dimensions. This requirement makes it impossible to grind chip breakers by the offhand method. It is best to do it with a universal cutter grinder or a small, reciprocating table, surface grinder of the types shown in (A) and (B) of Fig. 26. These machines have a

TABLE VII. PROPER WIDTHS FOR CHIP BREAKERS

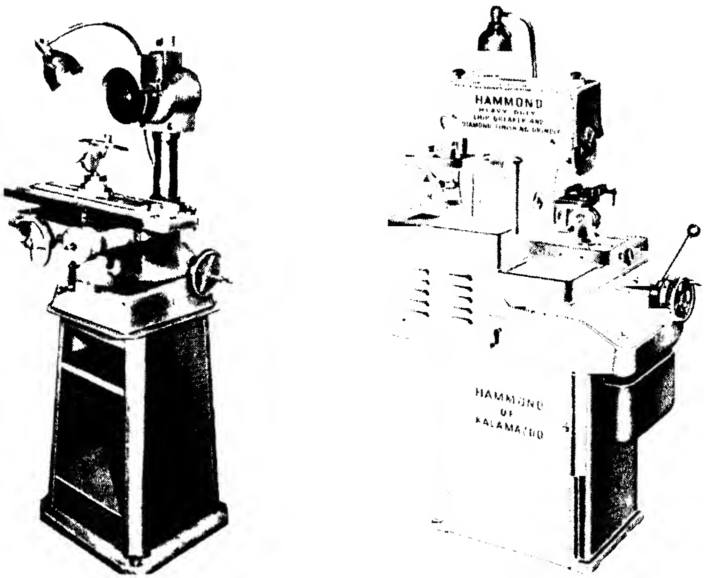
Cut Depth in Inches	Feed in Revolutions per Minute				
	.008-.012	.013-.017	.018-.022	.023-.027	.028-.032
1/64-3/64...	1/16	5/64	3/32	7/64	1/8
1/16-1/4....	3/32	1/8	5/32	11/64	3/16
5/16-1/2....	1/8	5/32	3/16	13/64	7/32
9/16-3/4....	5/32	3/16	7/32	15/64	1/4

universal vise fixture for holding and positioning the tool so that proper shape and width can be made certain. On either type of machine, the chip breaker is most easily and effectively ground with a resinoid-bound diamond wheel which is usually 4" to 6" in diameter, about 1/4" in width, and having one edge slightly rounded. The grit size of the wheel should be from 100 to 120. The vertical feed should be small—from .00025" to .0005" per pass. The table speed should be relatively fast and actuated by a hand crank.

Fig. 27 illustrates the setup for grinding chip breakers using a roller-type box tool which is used in turret lathe or screw machine work.

Sharpening Forming Tools. The turning of curved profiles is usually done with forming tools. The shape of the cutting edge of these tools is the identical relief counterpart to the generating line of the rotating body desired, the tool being fed into the work at right angles to

the direction of rotation. The sketch in Fig. 28 illustrates a forming tool having a complex profile, and the work performed by the tool. These tools are made with work clearance or relief angles just as were the other tools discussed in this chapter. However, forming tool profiles must be corrected for these angles. Such tools must be sharpened with great care, since even a little careless grinding can change their dimensional accuracy sufficiently to render the tool worthless. The right and the wrong way of sharpening forming tools is discussed here as



(A) (B)

Fig. 26. Two Types of Grinding Machines Used for Grinding Chip Breakers

(A) Courtesy of Kolbe & Sons Co

(B) Courtesy of Hammond Machinery Builders Inc

a means of helping the tool room man efficiently grind the tools in his care, and as a means of guiding the apprentice in the proper setup procedure. Forming tools, although they may have more than one cutting surface or are made with compound curves, still must be considered as single-point tools as opposed to such multiple-edge or bladed devices as milling cutters or reamers. They are, therefore, included in this section.

Flat Forming Tools. Forming tools of this type are shown in Fig. 29. The tool shown at (A) is similar to those used on lathes, planers, screw machines, special turning machines, and turret lathes. These tools are sharpened by grinding them across the top. The forming tools generally used for automatic screw machines and turret lathe

work are shown in (B) of Fig. 29. More detailed explanations of these tools will be given later. Meanwhile, it should be observed that the various rake and clearance or relief angles, while they may present peculiar problems in geometry to the tool grinder, serve the same purposes as the angles discussed earlier in the sections on straight, single-point cutting tools.

The straight tongue or dovetail forming tools may be made with the top straight as shown in (A) of Fig. 30, or with a back rake angle, N, as shown in (B). The clearance or relief angle, M, and the back rake angle, N, change the contour dimensions of the tool. In both (A) and (B) of Fig. 30, dimension Y is greater than dimension X, since Y is at right angles to the vertical center line of the work, while X is at right angles to the edge of the tool. Once the relationship between X and Y is established, it should be maintained throughout the life of the tool by sharpening on the top to the same relief and rake angles. If the sharpening is not carefully done, the relationship will change and the tool, more than likely, will be worthless.

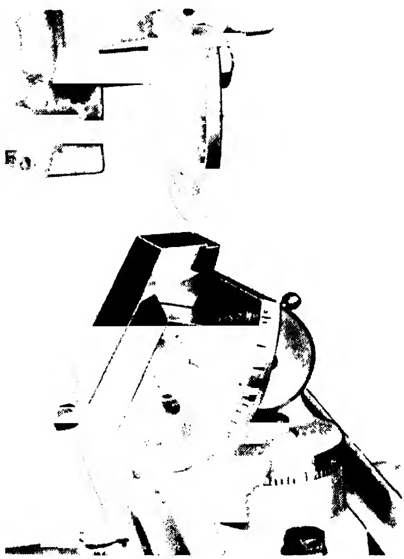


Fig. 27. Setup for Grinding the Chip Breaker
Courtesy of Kolbe & Sons Co

Circular Forming Tools. Some operators prefer the circular type of forming tool for automatic screw machine work and for turret lathe operations. An example of this tool is presented in Fig. 31. In this type of tool, the cutting edge is below the center line of the tool as shown by dimension h. When the tool is set with its cutting edge at the center of the work, there will be a clearance between the tool and the work as shown by angle a in (A). This angle is formed by two lines, one each of which is tangent to the work and the tool at the point of contact.

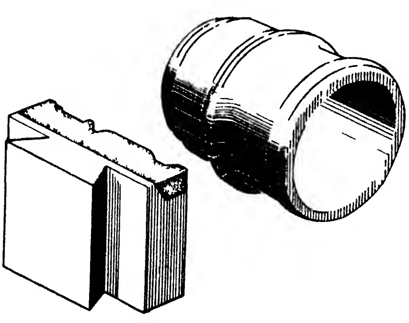


Fig. 28. Carbide Tipped Forming Tool and the Profile It Cuts

As seen in (A) of Fig. 31, the actual distance, Y, is greater than the difference between the radii R and r. Once these tools are cor-

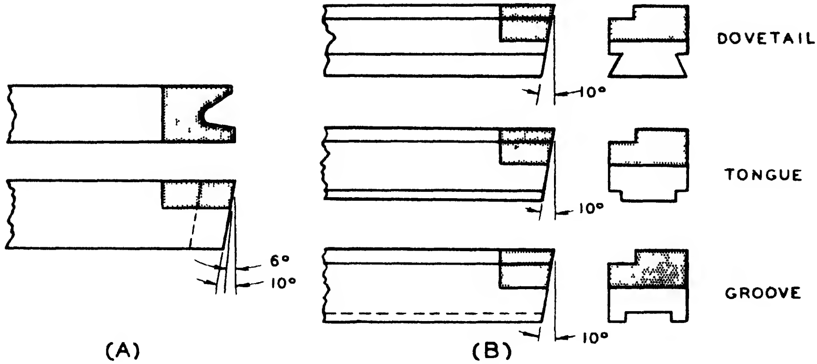


Fig. 29. Flat Forming Tools with Dovetails, Tongues, and Grooves

rectly made, the dimension Y should be maintained when regrinding becomes necessary. This is done by grinding so that when the tool is set in the machine at distance h above the center of the work, the face of the tool will coincide with the axis of the work as shown in (A) of Fig. 31.

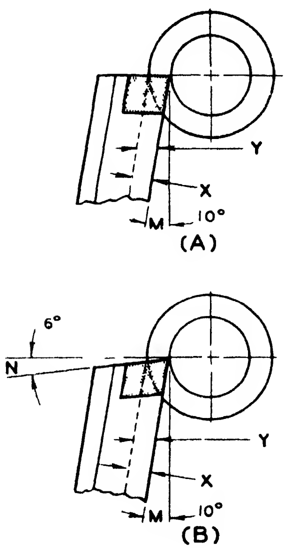


Fig. 30. Straight Forming Tool with Zero Back Rake (A) and with Positive Back Rake (B)

Circular forming tools are sometimes made with a positive back rake. In shop parlance, this is called a "hook." This back rake is illustrated in (B) of Fig. 31. Here, too, the cutting distance, Y, is greater than the difference between radii R and r. Once the hook angle has been established, it, likewise, should be maintained through careful sharpening.

Methods of Grinding. Flat forming tools should be held for sharpening in a universal vise or special vise fixture with the top in a horizontal position, as shown in Fig. 32 by line H-H. Altering this position to give the tool a back rake angle will change the cutting distance from ab to ac, which is greater than the dimension for which the tool was designed. Grinding the tool shown in (A), Fig. 32, to a negative rake angle would change the cutting distance, making it approximately equal to de on line T-T.

Straight forming tools with a back rake angle should be set for sharpening with the top in a horizontal position as shown in (B) of Fig. 32, thus inclining the tool to angle A + B, the sum of the front clearance and the back rake angles. If this is not done, the cutting distance will change in grinding.

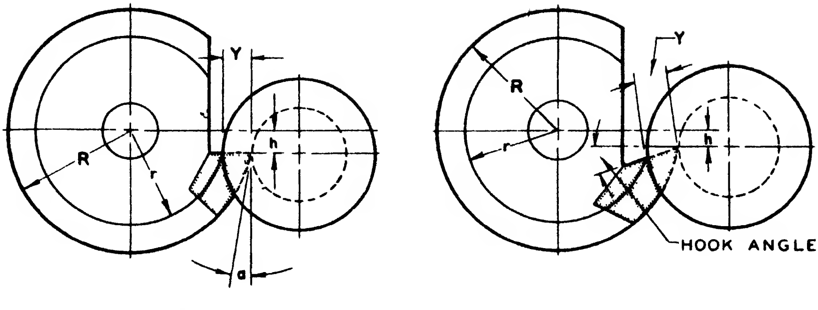


Fig. 31. Circular Forming Tool with Zero Back Rake (A) and with Positive Back Rake (B)

Circular forming tools with zero back rake should be ground so that distance h in (A) of Fig. 33 is not changed and the top face of the tool remains horizontal. This was shown in Fig. 31 at (A). Any regrinding along planes T - T or X -X, shown in Fig. 33, will change the cutting distance. Circular forming tools designed with a hook or back rake should be sharpened to the same angle. Otherwise, the cutting distance will change.

Selection of Wheels. Straight and circular forming tools should be sharpened to a fine finish. Other conditions remaining the same, the life of the tool between grinds and the degree of finish obtained on the work surface depends on the finish achieved in sharpening the tool.

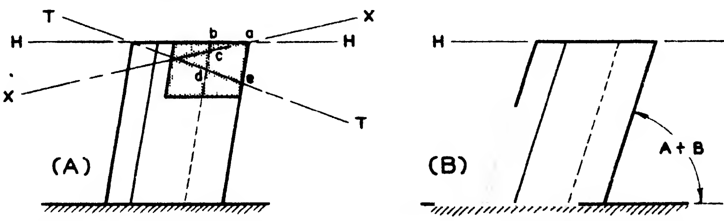


Fig. 32. Setting Straight Tools without Back Rake for Regrinding *(Left)* and Tool Which Has Back Rake *(Right)*

A straight wheel such as was shown in Fig. 1 at (A) should be selected for the rough grinding of forming tools. When the gash in the tool is narrow, the dish type of wheel shown in Fig. 34 is probably best. The grinding wheel grades usually chosen are F, G, and H. The grain size may be from 60 to 80 for roughing, since these sizes remove material relatively fast. Finish grinding should be done with 120-to 180-grain wheels. Wheels having this grain size result in a finer finish on the ground surface. For best results, no attempt should be made to remove more than .0005" per pass for rough grinding, and not more than .00015"

for finish grinding. A relatively slow table traverse should be used—about 25 to 35 f.p.m. Attempts to grind too much at any one pass may result in cracked cutting edges.

A number of examples of typical grinding setups commonly encountered in the toolroom are given in the following paragraphs. The proper way of holding the work in each case will be explained, together with data on the method that should be used.

Dovetail Flat-Top Tool. A setup is shown in Fig. 35 for sharpening a dovetail straight forming tool. The tool is held in the vise in such a way that its top is in a horizontal plane, or is said to be level. A straight grinding wheel is used both for finishing and roughing operations. When considerable material is to be removed, a coarse grinding

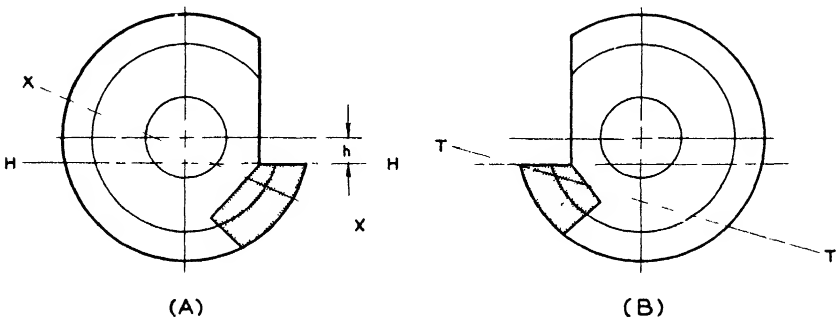


Fig. 33. How the Cutting Distance May Be Changed by Grinding Circular Forming Tools Incorrectly

wheel should be used. This wheel should be from 60 to 80 grain size. Either a finer wheel (100 to 180 grain size) should be used for finishing or the tool lapped on a diamond wheel.

Flat Dovetail with Back Rake. In Fig. 36 is illustrated a setup for sharpening a straight dovetail tool in which a vise is used. The use of the vise permits the setting of the tool to the clearance and back rake angles by means of the graduations on the vise dial, details of which were shown in Fig. 20. When the tool has front clearance and a back rake angle, it should be set for sharpening to the sum of the two angles. For example, a tool having a 10° clearance angle and a 6° back rake angle should be set for sharpening to 16°. This is shown in (B) of Fig. 32.

Circular Tool with Zero Back Rake. Fig. 37 shows the setup for sharpening a circular forming tool with zero back rake. This tool has an opening of sufficient size to permit the use of a straight grinding wheel. The tool is set in a vise as shown. The vise may be clamped to the table or held on a magnetic chuck. It is necessary to set it so that after grinding, distance h, as shown in (A) of Fig. 31, is maintained. Where there are a number of such circular forming tools to be

sharpened, a simple bracket fixture may be used for holding the tool. Such a bracket is illustrated in Fig. 38.

Circular Tool with Positive Back Rake. Circular forming tools with positive back rake can be set for sharpening in a vise or special grinding fixture as shown in Fig. 39. A dish-type wheel can be used whenever the opening is large enough to permit it. The tool is set with its cutting face in a horizontal position. The tool is then advanced for grinding by rotating it slightly as indicated by the arrow in Fig. 40.

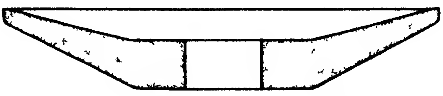


Fig. 34. A Dish-Type Grinding Wheel

Inspection After Sharpening. Once circular forming tools have been sharpened, they should be inspected for keenness of cutting edge, smoothness of the finish obtained, distance h, and back rake angle. Recommended offsets are given in Table VIII. Flat forming tools should be inspected for front clearance or relief angle, keenness of the cutting edge, back rake, and the smoothness of the finish.

When all other conditions remain the same, the smoothness of finish, particularly on the front face of the tool, has a considerable effect on the smoothness of the work produced and on the life of the tool. The smoothness of the top of the tool has a great influence on its performance. For best results, the tool should be honed after grinding with an extremely fine diamond hone of the type shown in Fig. 41.

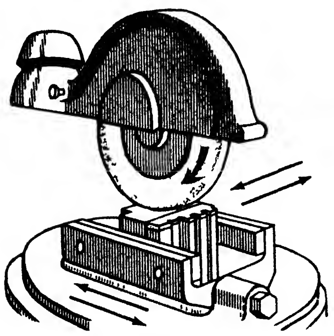


Fig. 35. Sharpening a Straight, Dovetail Forming Tool

Grinding Summary. As a means of emphasizing some of the more important points covered in this chapter, this summary is presented for the further guidance of the operator.

1. Carbide tools should be ground with care so as to avoid overheating of the tip. Overheating is likely to check or crack the tip.

2. Grinding may be done either wet or dry. When grinding wet, a sufficient amount of cooling liquid should be used to keep the work cool. Intermittent cooling and heating of carbide tips during grinding will result in cracking of the tips.

3. The hot tool should never be dipped in a coolant. Such action may crack the tip. Natural, slow cooling in air is preferred.

4. Offhand grinding, whenever practical, is preferred to machine grinding. There is more flexibility in hand grinding than in that done by machines. With machine grinding, unless extreme care is exercised, more cracked tool tips can be expected than from careful, expert hand grinding.

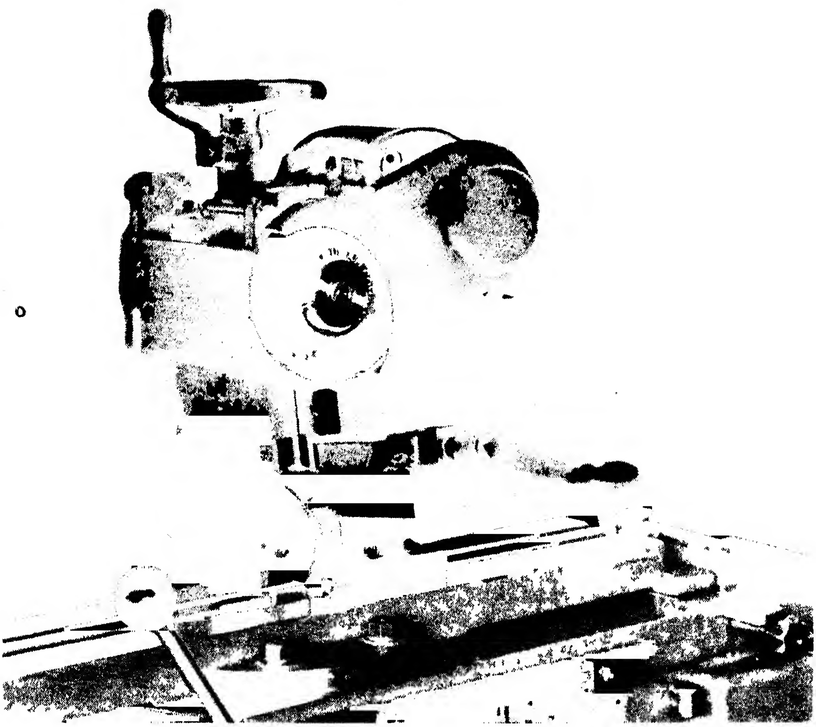


Fig. 36. Sharpening a Straight, Dovetail Forming Tool with Back Rake

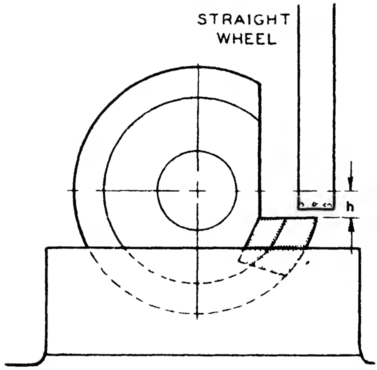


Fig. 37. Grinding a Circular Forming Tool Held in a Vise

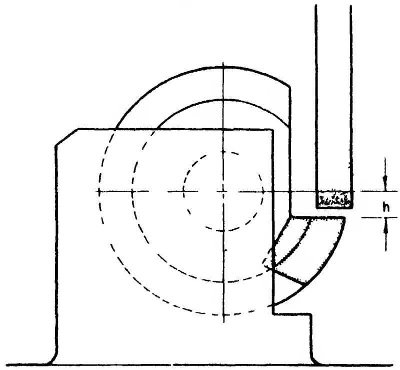


Fig. 38. Grinding Fixture for Circular Forming Tools

TABLE VIII. RECOMMENDED OFFSETS FOR STRAIGHT FACE CIRCULAR FORMING TOOLS

Machine	Machine Size or Capacity	Recommended Large Radius of Tool in Inches	Offset in Inches
Acme	51 and 515	.750	.0937
	52	1.000	.937
	53	1.875	.125
	54 and 55	1.250	.156
	56	1.500	.1875
Brown and Sharpe	00	.875	.125
	0	1.25	.156
	2	1.50	.250
	6	2.00	.312
Cleveland	1/4	.625	.031
	3/8	.843	.0625
	5/8	1.562	.0625
	7/8	1.875	.0625
	1 1/4 and 2	1.375	.0625
	2 1/4	1.625	.0625
	2 3/4 and 3 1/4	1.875	.156

Fig. 39. Grinding a Positive, Back Rake, Circular Forming Tool, Holding It in a Special Fixture

Courtesy of the Norton Company

5. In machine grinding, never more than .001" should be fed per pass for roughing operations, and never more than .0005" per pass for finishing operations.

6. In hand grinding, the wheel should be dressed frequently, the tool being pressed gently against the face of the wheel as it is moved continuously from side to side.

7. Tools should be reground before they become extremely dull.

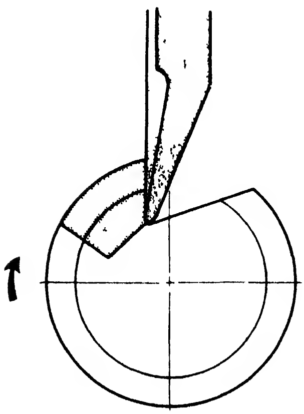


Fig. 40. Advancing a Circular Forming Tool with Positive Back Rake for Grinding

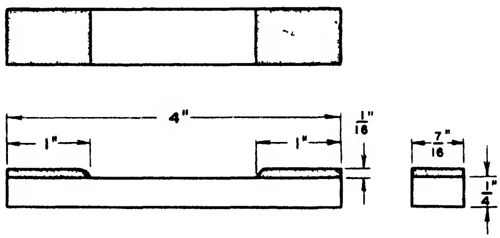


Fig. 41. Fine Diamond Hone for Touching Up Tool after Grinding

This is an economical practice, for then it is only necessary to grind off a small amount of material in order to recondition the tool.

8. In sharpening tools, the proper rake and clearance angles should always be maintained as recommended by the tool engineering department or the tool manufacturer.

9. The featheredge of the tool should be honed slightly before it is used. This procedure will pay dividends in longer tool life. The carbide tip should be touched up with a diamond hand hone between grinds, as this will maintain a keen cutting edge and will add to tool life.

10. The grinding wheel should always be well guarded. Safety goggles for the production of the eyes should be worn at all times when working around any grinding wheel.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading The important points in the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Define the grit and the grain size of an abrasive.

2. State the difference, if any, between the grit and the grain size.
3. Name the abrasives commonly used in grinding wheels for sharpening sintered carbide tools.
4. Define the grade of a grinding wheel.
5. Define the structure of a grinding wheel.
6. Define the cutting speed of a grinding wheel.
7. In what units of measure is the cutting speed expressed?
8. Define the feed per pass for the grinding wheel.
9. Explain the meaning of the symbols in the following wheel marking: 20-C-80-H5-B-10.
10. An 8'' diameter grinding wheel is to run at 4,000 f.p.m. The speeds available on the machine are 2,000, 2,500, and 3,000 r.p.m. Which machine spindle speed would give the nearest specified peripheral speed?
11. A grinding machine has three spindle speeds of 1,200, 1,400, 1,650 r.p.m. How many r.p.m. should the spindle make to put the cutting speed of a grinding wheel at 2,200 f.p.m.?
12. Find the cutting speed of a diamond wheel which is 6'' in diameter and revolving at 3,500 r.p.m.
13. Find the cutting speed of a grinding wheel which is 7'' in diameter and revolving 1,650 r.p.m.
14. What is the cutting speed of a wheel 1/2'' in diameter which is revolving at 20,000 r.p.m.?
15. What diameter wheel should be put on a grinder spindle making 3,580 r.p.m. in order that the cutting speed be 7,500 f.p.m.?
16. What is the value for h on a 3'' straight-faced circular forming tool to be used in an Acme #53 automatic screw machine?
17. Why is the circular forming tool cutting face made so that it is below the center line of the tool?
18. Can the circular forming tool made without back rake be reground so as to give it positive or negative back rake?
19. What should be the width of a chip breaker for .020'' per revolution feed at a 1/2'' cut depth?
20. What would happen if soft steel were being cut at 500 f.p.m. with a tool having no chip breaker?

REFERENCES

Brown and Sharpe Mfg. Co. "Construction and Use of the #10 Cutter and Grinding Machine," Providence, R.I.

Carborundum Co., The. "Diamond Wheels," "Grinding Cemented Carbides with Green Grit Wheels."

Johnson, Haywood. Grinding Wheels and Their Uses. Cleveland, Ohio: Penton Publishing Co., 1942.

LeBlond, R. K., Machine Tool Co. Manual of Tool Room Grinding.

Norton Company. Grinding Carbide Tipped Tools, Grinding Wheel Data Book.

CHAPTER VI

Cutting Power, Speeds, Feeds

Cutting Speed. It was carefully pointed out in the first chapter that the cutting speeds and feed rates not only possible but actually in use constitute the crux of the carbide "revolution" in industry. This chapter expands that thought and contains a considerable amount of information in text and table form, based on original research concerning cutting speeds and feeds. However, it should be explained that this data by no means represents the last word on the subject. Research projects are now under way in widely separated, well-known universities and in the industries. The purpose of this extensive study is to set limits on speeds and feeds and to form definitive recommendations. Thus far, because of the newness of the subject and the continual and ever more startling discoveries made daily, these projects have been able to do little more than indicate trends. In view of these conditions, only information which is safe and certain is presented in this chapter. What is known of speeds and feeds in connection with the carbides will be discussed, as well as the forces required to cut various materials under diverse conditions, and the machinability ratings of many metals and alloys.

The cutting speed in turning, boring, and facing with a single-pointed tool is the rate at which a point in the circumference of the work passes the point of the tool. The cutting speed is expressed in surface feet per minute (f.p.m.) and may be found by the following formula:

$$S = \frac{3.14 \times D \times N}{12}$$

S = the cutting speed in feet per minute or f.p.m.
D = the diameter of the work in inches
N = the revolutions per minute (r.p.m.) of the work
3.14 and 12 are the common conversion constants

To illustrate the use of this formula, assume it is desired to find the cutting speed of a piece of work which is 6" in diameter and is revolving at 200 r.p.m. Substituting the known values for letters reresults in

$$S = \frac{3.14 \times 6 \times 200}{12} = 314 \text{ f.p.m.}$$

Cutting speeds for given materials are usually known. They may be obtained from tables included in this book, from any of the various machinists' handbooks, from the literature of manufacturers of machinery, or from tool manuals of operation. It is usually necessary for the mechanic and the tool engineer to find the number of r.p.m. the work should make and then set the machine to this most economical speed. To figure r.p.m., a second formula is used which is derived from the first by simple transposition.

$$N = \frac{S \times 12}{3.14 \times D}$$

The symbols in this formula have the same value as in the first formula.

To find the r.p.m. a 5'' shaft should make in order that its surface may be machined at 350 f.p.m., the known values are substituted for the symbols resulting in

$$N = \frac{250 \times 12}{3.14 \times 5} = 191 \text{ r.p.m.}$$

Table I presents a number of cutting speeds and work diameters and the resulting r.p.m.

Cutting Feeds. The term cutting feed, or simply feed, is used to express the axial distance the tool moves in each revolution of the work. It may be used also by some machinists and engineers to des-

TABLE I. CUTTING SPEEDS AND REVOLUTIONS PER MINUTE

Diameter of work in inches	Cutting speed in feet per minute									
	25	50	70	100	150	200	250	300	350	400
	Revolutions per minute									
1/2........	191	382	535	764	1146	1528	1910	2292	2674	3056
3/4........	127	254	357	508	762	1016	1270	1524	1785	2032
1..........	95.5	191	267	382	573	764	955	1146	1335	1528
1 1/2.....	63.7	127	178	255	381	510	635	762	890	1020
2..........	47.8	95.5	134	191	286.5	382	477.5	573	670	764
2 1/2.....	38.2	76.3	107	153	229	306	381.5	458	535	612
3..........	31.8	63.7	89.1	127	191	254	318.5	382	445.5	508
3 1/2.....	27	54	76	109	162	218	270	324	380	436
4..........	28.5	47	66	95	141	190	235	282	330	380
4 1/2.....	21	42	59	84	126	168	210	252	295	336
5..........	19	38	53	76	114	152	190	228	265	304
5 1/2.....	17	34	48	69	102	138	170	204	240	276
6..........	15.5	31	44	63	93	126	155	186	220	252
6 1/2.....	14.5	29	41	58	87	116	145	174	205	232
7..........	14	27	38	54	81	108	135	162	190	216
8	11 5	23	33	47	69	94	115	138	165	188

TABLE II. CLASSIFICATION OF CUTS

Type of Cut	Depth of Cut	Cutting Speed
Heavy roughing..................	3/8 or more	slow
Medium roughing..............	3/16 to 3/8	medium
Light roughing..................	1/32 to 3/16	fast
Heavy finishing..................	1/16 to 1/8	slow
Medium finishing...............	1/32 to 1/16	medium
Light finishing..................	1/64 or less	fast

ignate chip thickness or chip load. The feed varies with the kind of cut taken and the finish desired. A coarse feed is usually used for roughing and a fine feed for finishing operations. The feed for sintered carbide tools used on roughing cuts may be from .015'' to .062'', depending upon

TABLE III. CUTTING SPEEDS AND FEEDS FOR VARIOUS MATERIALS MACHINED WITH CARBIDE IN A TURRET LATHE

Material Cut	Cutting Speed		Feeds			
			Ram-Type Machine		Saddle-Type Machine	
	Roughing	Finishing	Roughing	Finishing	Roughing	Finishing
Cast iron	180-200	350-400	.016-.025	.016-.025	.032-.060	.032-.125
Semisteel (hard)	140-160	250-300	.016-.025	.016-.025	.032-.060	.032-.060
Malleable* cast iron	250-300	400-500	.016-.025	.016-.025	.032-.060	.032-.060
Steel casting* 35% carbon	150-180	200-250	.010-.020	.012-.020	.018-.030	.018-.045
Brass (commercial 85-5-5)	600-1000	600-1000	maximum	maximum	maximum	maximum
Bronze* (80-10-10)	600	1000	.016-.030	.010-.016	.016-.050	.010-.030
Aluminum†	800	1000	fine	fine	fine	fine
S.A.E. 1020* (coarse feed)	300	300	.020-.030	.020-.030	.024-.044	.024-.044
S.A.E. 1020* (fine feed)	450	450	.077-.020	.007-.020	.010-.030	.010-.030
S.A.E. 1035*	250	250	.020-.030	.020-.030	.024-.044	.024-.044
S.A.E. 1315*	400-500	400-500	.010-.015	.010-.015	.010-.015	.010-.015
S.A.E. 1050*	200	200	.012-.020	.012-.020	.025-.090	.015-.045
S.A.E. 2315*	300	300	.012-.020	.012-.020	.025-.090	.015-.045
S.A.E. 3150*	200	200	.012-.020	.012-.020	.025-.090	.015-.045
S.A.E. 4150*	200	200	.012-.020	.012-.020	.025-.090	.015-.045
Stainless steel (selenium)	240-300	240-300	.010-.015	.010-.015	.010-.015	.010-.015

*Water soluble oil lubricant.
†Kerosene lubricant.

Courtesy of Warner & Swasey Co.

the available power and the rigidity of the work and the tool. The feed for finishing cuts is usually from .004" to .025" per revolution, depending on the degree of finish desired.

Roughing and Finishing Cuts. Machining cuts are commonly classified as rough, semi-finish, and finish. This classification, however, does not necessarily mean the relative results obtained from the various cuts. In addition, these terms have different meanings in different industries. In the heavy machine industries, for example, a roughing cut has a different meaning than in the light industries in which the same term is used with regard to the processing of small parts.

Cutting speeds for turning tools depend on many factors such as the properties of the material being cut, the tool material, the shape and size of the tool, the tool life that is desired, the depth of cut, the feed, the coolant used, the rigidity of the work, etc. The use of a proper coolant often results in an increase of cutting speed up to 25 per cent for the same tool life.

TABLE IV. SUGGESTED TOOL ANGLES, CUTTING SPEEDS, AND CUTTING FLUIDS FOR TURNING VARIOUS METALLIC MATERIALS WITH CARBIDE

Material Cut and Hardness	Ultimate Tensile Strength in 1,000 p.s.i.	Suggested Tool Angles in Degrees			Cutting Speed Range in f.p.m.	Cutting Fluid Used
		Side Rake	Back Rake	Side and End Relief		
Cast iron						
130-175 Brinell	18-30	10	0-4	4	275-400	dry
175-200 Brinell	20-35	8-10	0-4	4	250-350	dry
alloy, 200-225 Brinell	30-36	4-6	0	4	150-250	dry
malleable	35-45	10	6	5	175-250	emulsion
S.A.E.						
1112 screw stock	70-90	10-20	6-10	5-8	300-400	sulphurize mineral oi
1120 " "	70-85	10-15	5-7	4-6	300-400	"
1020 soft forging	63-80	10-15	5-7	4-6	300-400	"
1030	72-86	10-15	6	4-6	250-350	"
1050	80-100	10-15	0	4-6	200-300	"
2315	80-115	10-15	6	6	200-275	"
3120	80-110	10	6	6	175-200	"
3140	90-115	10	6	6	150-200	"
52100	100-125	10	0	6	150-200	"
6150	125-150	10	0	6	125-175	"
Stainless steel	85-110	10-15	6	5	150-200	"
Structural steel						
170-200 Brinell	50-70	6-10	0	6	175-250	"
Aluminum	19	10-20	20-40	10	500-100	kerosene
Rolled copper	30	18-25	0	10	300-500	dry
Yellow cast brass	25	0	0	6-8	300-500	dry
Bronze, cast	30	6-10	0	6	250-400	dry
Phosphor bronze	25	4-8	0	6	150-300	dry

TABLE V. SUGGESTED CUTTING SPEEDS, FEEDS, AND DEPTH OF CUT FOR MACHINING FERROUS CASTINGS WITH CARBIDE IN ENGINE OR TURRET LATHE

Material Cut and Brinell Hardness Number	Cutting Speed in Feet per Minute (Roughing)			Cutting Speed in Feet per Minute (Finishing)		Cutting Fluid Used
	Depth 3/8-up Feed .025-.062	Depth 3/16-3/8 Feed .0156-.032	Depth 3/32-3/16 Feed .010-.025	Depth 1/64-3/32 Feed .006-.015	Depth .004-.015 Feed .004-.010	
Cast iron						
plain						
100-130	225	250	275	300	400	dry
131-160	200	225	250	275	350	"
161-200	175	200	225	250	300	"
201-230	150	175	200	225	275	"
alloy						
175-220	175	200	225	250	275	"
221-260	150	175	200	225	250	"
267-300	125	150	175	200	225	"
301-350	100	125	150	175	200	"
chilled						
325-350	100	125	150	175	200	"
351-400	75	100	125	150	175	"
401-450	50	75	100	125	150	"
451-500	25	50	75	100	125	"

malleable (plain) 100-125	200	225	250	300	325	emulsion
126-150	175	200	225	275	300	"
(pearlitic) 130-175	175	200	225	250	275	"
176-212	150	175	200	225	250	"
213-255	125	150	175	200	225	"
256-300	100	125	150	175	200	"
310-350	75	100	125	150	175	"
(alloy) 130-165	175	200	225	250	275	"
166-200	150	175	200	225	250	"
201-235	125	150	175	200	225	"
brake drums		200	225	250	300	dry
semisteel (25% steel, 2% si)	200	220	240	260	300	"
(30% steel)	160	180	200	220	260	"

Roughing cuts are made so as to obtain the most economical removal of material from the part being machined. This calls for a correlation of cutting speed, feed, and depth of cut so as to bring about the desired finish while maintaining tool life at the practical maximum. Roughing cuts are taken at slower speeds than are the finishing cuts. They are carried on with heavy feeds up to the limit of the machine's power, tool strength, and the strength of the work itself.

Finishing cuts are taken at higher speeds, with finer feeds, and with shallower depths of cut. Soft materials give best results at higher speeds when the depth of cut is from .010" to .015". Hard materials will give more satisfactory results with a cutting depth from .004" to .010" and with a fine feed.

In tooling up, cutting conditions should conform to the best practice based on past experience. Later the setups may be modified to obtain better results and more efficient performances. In general, roughing and finishing cuts are necessary for the removal of large amounts of material from machined parts. A rough cut, a semifinish cut, and a finish cut may be necessary for work requiring the utmost accuracy.

Commercial Cutting Speeds. Commercial cutting speeds, unlike those used in test laboratories, are based on tool life more than anything else. If tool life can be extended two to five hours between grinds, factory efficiency is greatly increased. In many cases, the life of the tool may be more important than the maximum amount of material per tool grind. In such cases, the operating speed and feed should be so adjusted as to give maximum tool life. Again, a tool that is difficult to grind because of its shape, or is difficult to set up in the machine, or a tool that is intended to produce many pieces within a certain limit of accuracy should be operated at such a speed as to permit long life between grinds.

The roughing tool, the function of which is to remove a maximum amount of material, may be reground more often. The finishing tool, required to turn out many pieces of the same size or finish, should be used in such a way that grinding is called for less often.

In rough turning with carbide tools, the most economical results are obtained when the speed, the feed, and the depth of cut are so correlated as to cause the tool to be resharpened at a definite time. The prevailing practice in many plants is to remove the tool for resharpening whenever it has finished a given number of parts. Under such conditions the tools are standardized as to size and shape, and the grinding is done by a specialist on machines made for the purpose.

Cutting speeds and feeds for turning a variety of materials on a turret lathe are suggested in Table II. These speeds and feeds may be for lighter cuts and lower for heavier cuts. The data in Table II should be used as a guide only, and should be changed to meet the various conditions of tools and materials as they are found to exist.

Table III gives suggested speeds and feeds for various materials machined in a lathe with single-pointed sintered carbide tools. Again, this table should be used only as a guide.

Table IV suggests cutting speeds, cutting fluid, and tool angles when machining metallic materials. This data is based on the hardness of the material cut and should be used only as a guide in setting up a job.

Table V gives cutting speeds, feeds, and depths of cut suggested for machining ferrous castings in an engine lathe or a turret lathe. These speeds are based on the hardness of the material cut and will be of value as a reference in setting up a job.

Table VI gives suggested cutting speeds for materials which are nonmetallic.

TABLE VI. CUTTING SPEEDS FOR MACHINING NONMETALLIC MATERIALS WITH CARBIDE

Material Cut	Cutting Speed in Feet per Minute	
	Roughing	Finishing
Asbestos	200-500	300-600
Bakelite	200-400	400-600
Carbon	300-700	400-600
Casein products	300-500	400-600
Concrete	100-300	200-500
Fiber	300-400	400-600
Glass	75-100	100-150
Hard rubber	200-300	300-500
Linoleum	200-400	300-600
Masonite	300-500	400-600
Mica	200-500	400-600
Porcelain, unfired	300-500	400-600
Tile	100-300	300-500
Wood	300-1000	500-1500

Cutting Time Requirements. The actual time required to make a cut in turning, boring, and facing operations may readily be determined by using this formula:

$$T = \frac{L}{f \times N}$$

T = time in minutes
L = length of cut in inches
f = feed in inches per revolution
N = work speed in revolutions per minute

To demonstrate the use of this formula, assume it is desired to determine the cutting time for a shaft which is 6'' in diameter, 4'0'' in length, and which is revolving at a speed of 200 r.p.m. If the feed is .016'' per revolution, the known values, substituted for letters, result in

$$T = \frac{48}{.016 \times 200} \text{ or 15 minutes}$$

The time required for a facing operation is figured in the same manner as for turning operations except that the difference in the radii of the work is substituted for L (length of cut in inches). Assuming the larger radius of a piece of work to be 3", the smaller radius 1", the feed .012" per revolution, and the work turning at 200 r.p.m., the time required to make the facing operation is

$$T = \frac{2}{200 \times .012} = 0.83 \text{ minute}$$

Tool Life and Cutting Speed. In cutting any given material, tool life is dependent primarily on the speed employed. Other factors affecting the life of the tool are feed, depth of cut, use of coolant or lubricant, the condition of the tool, the tool holding device, and the machine. It is assumed in the following formulas and examples that these and other variables such as the properties of the material being machined, remain constant.

The relationship between cutting speed and tool life between grinds for a given tool, material, feed, and depth of cut is expressed by this formula:

$$VT^n = C$$

V = the cutting speed in f.p.m.
T = tool life in minutes under actual cutting conditions
^{n} = exponent, depending on material cut and tool shape
C = constant, depending on conditions and material cut, representing cutting speed, one minute of tool life

The values of C, ^{n}, and T are found by experimentation. For example, when cutting steel with sintered carbide tools, the value of ^{n} has been found to be approximately 0.16. The value of C varies with the feed taken and the depth of cut used. For estimating purposes, the value of this factor can be taken as 600 for heavier cuts and about 800 for lighter cuts.

To illustrate the application of this formula, assume it is desired to discover at what speed machining should be done when cutting S.A.E. 1050 annealed steel at .025" feed per revolution, and 3/16" depth of cut. C is calculated at 600 and the desired tool life is 180 minutes.

The known factors are ^{n} = .16, C = 600, and T = 180. Substituting these values in the formula results in

$$V \times 180^{.16} = 600$$

Solving algebraically for V results in

$$V = \frac{600}{180^{.16}}$$

Working with such large fractional exponents will be found much easier when logarithms are used in the computations. Therefore, V = log 600 – (log 180 × .16) or 2.77815 – (2.25527 × .16) = 2.41731. V is obtained by finding the number, the log of which is 2.41731. This number is 261.4 and means that the cutting speed for a tool life of 180 minutes should be 261 f.p.m.

In a second example on the use of the formula for determining cutting speed with relation to tool life, assume the cutting speed is to be found where only a 90-minute tool life is required. All other conditions are as previously given. Substituting the values for the letters in the original formula gives

$$V \times 90^{.16} = 600$$

$$V = \frac{600}{90^{.16}}$$

$$V = \log 600 - (\log 90 \times .16)$$

$$V = 2.77815 - (1.95424 \times .16)$$

$$V = 2.46537$$

$$V = 292 \text{ f.p.m.}$$

Tool life refers to the actual cutting time, exclusive of the time required to load, unload, and measure the work. The ratio of cutting time to the total time necessary to complete a piece of work is generally one to four. Under these conditions, a tool, the life of which was estimated at 90 minutes, could be kept on the job a total of 360 minutes (90 × 4), or six hours. Under special conditions, tools may be in use for a larger percentage of the total time. In a situation of this nature, it is obvious that the tool must be resharpened more frequently unless the speed is adjusted downward.

There are records of carbide tools, which, when properly ground and set and when operating at relatively low speeds on brass, have done satisfactory work for more than four months, being in use 16 hours daily, seven days per week! This represents a tool life of better than 1,900 hours between grinds.

Machinability of Metals. Machinability testing is an exact scientific comparison of the materials to be cut under accurate control conditions. This calls for time to prepare the samples and the tools, and to make the setups. As might be expected, it is necessary to use identical tools in testing. Otherwise, misleading results would be obtained. The tools should be accurately ground to gage and set to precision. Cutting edges should be carefully honed so as to remove any loose material that might adhere to the cutting edge.

Comparative machinability observations have been recorded ever

TABLE VII. RELATIVE MACHINABILITY OF STEELS

Cold-Drawn and Hot-Rolled				Hot-Rolled and Annealed			
S.A.E. Number	Condition	Approximate Brinell Hardness	Machinability Rating	S.A.E. Number	Condition	Approximate Brinell Hardness	Machinability Rating
1010	cd	131-170	42	1095	annealed	190-220	45
1015	cd	135-170	50	1330	"	179-235	50
1020	cd	137-174	65	1340	"	179-235	45
1025	cd	160-200	65	2320	hr	175-220	50
1030	cd	170-212	65	2330	hr	179-235	45
1035	cd	175-217	60	2340	annealed	179-235	45
1040	cd	175-217	68	2350	hr	190-240	45
1112	cd	179-229	100	2515	hr	175-220	47
X1112	cd	179-229	135	3120	hr	140-160	50
1120	cd	179-229	80	3130	hr	185-220	45
X1315	cd	143-179	80	3140	annealed	187-229	55
2315	cd	174-220	55	X3140	"	187-229	55
3120	cd	163-206	60	3150	"	200-240	50
4615	cd	175-217	65	3250	"	195-230	44
5120	cd	117-212	65	X4130	"	187-229	65
6120	cd	180-218	50	4140	"	190-235	56
1015	hr	110-130	42	4150	"	187-235	50
1020	hr	130-150	48	X4340	"	220-245	58
X1020	hr	135-160	62	4620	hr	165-195	58
1025	hr	130-150	58	4640	annealed	187-235	55
1030	hr	135-150	60	4820	hr	190-220	55
1035	hr	160-180	55	5140	annealed	174-229	60
1045	hr	180-220	55	52100	"	183-229	30
1085	hr	185-220	48	6130	"	210-225	55
				6145	"	179-235	50

since carbides were introduced. These figures have been averaged to within five per cent from various users' records. They are based on a 100 per cent rating for S.A.E. 1112 cold-drawn or cold-rolled Bessemer steel, machined under normal cutting conditions and using a suitable cutting fluid. These machinability ratings are given in Tables VII and VIII. They take into consideration such factors as the type of machining, speed, and power used, etc. When a rating of 50 per cent appears with the material rated, it is an indication that the material has a general, over-all machinability approximately half that of S.A.E. 1112. It means greater difficulty will be encountered in machining it with consequent shorter tool life, poorer surface finish, and increased power consumption. Generally speaking, when machinability is lower, the speed of the cutting action is decreased also.

As in previous tables, the figures given in Tables VII and VIII do not represent absolute values. The information should serve only as a guide for use under average shop conditions.

Machinability of Alloy Steels. Table VII indicates that machinability ratings for alloy steels are lower than that for S.A.E. 1112

steel. They are, therefore, more difficult, in general, to machine, although not necessarily so much so as the hot-rolled, low-carbon and higher-carbon steels. Low-carbon and medium-carbon steels are soft and are difficult to machine for that reason. For best performance in

TABLE VIII. RELATIVE MACHINABILITY OF NONFERROUS METALS

Material	Machinability Rating	Material	Machinability Rating
Aluminum		Copper	
11-S	500-2000	cast	70
2-S	300-1500	rolled	60
17-S	300-1500	Gun metal	60
Brass		Inconel	45
leaded	150-600	Monel metal	
red	200	cast	35
yellow	200	rolled	45
Bronze		K monel	50
lead bearing	200-500	Magnesium alloys	500-2000
manganese	40	Nickel	20
Everdur	60	Zinc	200

machining, low-carbon steels require tools with keener cutting edges than do alloy steels. In addition, tools used for machining alloy steels usually have larger lip angles and a small side rake.

Hardness and Machinability. The hardness of the metal cut is not necessarily an index of machinability. Alloy steels having the same Brinell hardness number as straight-carbon steels usually have a lower machinability rating. The reason is that they are stronger, and the chip from them exerts more pressure against the tool surface than that from straight-carbon steel. Usually, too, more heat is generated in the cutting action.

It is a commonly accepted fact that steels softer than 180 Brinell machine with some degree of difficulty because of the softness of the material. On the other end of the scale, steels of greater hardness than 400 Brinell also machine with difficulty, while steel of 500 to 600 Brinell machine with even greater difficulty. When working with the harder steels, exacting conditions of application are required along with, usually, the use of negative rake angles.

Table IX gives cutting speeds for roughing and finishing cuts based on the hardness of the material cut. These values are fairly accurate for straight-carbon steels but are not too reliable for alloy steels where metallurgical conditions play an important role in the cutting action. The values given in the table should be used with discretion in the selection of economical speeds based on the hardness of the materials cut.

Machinability of Cast Metals. The machinability of cast iron and of malleable cast iron is good. Castings, when free from hard spots

and sand inclusion, allow long tool life between grinds, permit heavy cuts to be taken, and result in good finish, while demanding relatively small power requirements.

TABLE IX. SUGGESTED CUTTING SPEEDS FOR SINTERED CARBIDE TURNING TOOLS *

Brinell Hardness Number	Surface Speed in Feet per Minute	
	Roughing cuts 1/2"-1/8" in depth	Finishing cuts 1/8"-.008" in depth
175	350-475	400-530
200	315-400	350-460
225	270-340	300-400
250	230-300	270-350
275	200-250	230-300
300	170-225	200-260
325	140-195	170-225
350	120-170	140-195
375	100-140	120-165
400	80-120	100-140

*Based on hardness for feeds from .006" to .032".

Cutting Forces. For a given material being turned, bored, faced, planed, or shaped, the cutting forces involved may be resolved into the tangential, or the down pressure on the tool; the radial, or the force tending to push the tool away from the work; and the longitudinal, or the force necessary to push the tool into the material. These cutting forces vary with the shape of the tool and the material being cut. They remain practically constant throughout the life of the tool until the limit of endurance is reached and the tool fails. At this point there is an increase in pressure and power consumption.

The tangential force, or the chip pressure on the tool, depends on the material being cut, the shape and sharpness of the tool, the depth of the cut, and the rate of feed used. This pressure or force on the tool is expressed by the formula

$$P = K \times f^{y} \times d$$

P = pressure on the tool in pounds
K = a material constant, also an experimental value
f = the feed in inches per revolution
y = the experimental exponent which varies with the shape of the tool and the material cut
d = the depth of cut in inches

A more simple formula which is sufficiently accurate for most shop conditions is

$$P = K \times f \times d$$

P = chip pressure on the tool in pounds
K = the material constant
f = feed in inches per revolution
d = depth of cut in inches

To illustrate the use of this formula, assume it is desired to determine the chip pressure on the tool in cutting high-carbon steel when the feed is .012" per revolution and the depth of cut 1/8". The average values for the constant, K, can be taken from Table X. The solution consists merely of substituting the known values for the letters of the formula.

$$P = 340{,}000 \times .012 \times .125 = 510 \text{ pounds}$$

TABLE X. K VALUES FOR CUTTING WITH SINGLE-POINTED TOOLS

Material Cut	Constant K (p.s.i.)
Mild steel	270,000
Medium-carbon steel	300,000
High-carbon steel	340,000
Low-alloy steel	350,000
Medium-alloy steel	375,000
Wrought iron	198,000
Cast iron	132,000
Bronze	110,000

Horsepower Required. The horsepower required at the point of the tool to cut the material is equal to the chip pressure on the tool, multiplied by the cutting speed in f.p.m., divided by 33,000 (the number of foot-pounds in one horsepower). This results in the formula

$$hp = \frac{P \times S}{33{,}000}$$

hp = horsepower required to make the cut
P = chip pressure on the tool in pounds
S = cutting speed in feet per minute
33,000 = foot-pounds of mechanical work done in one minute

Assume it is desired to find the horsepower necessary to cut the material at a cutting speed of 300 f.p.m. under the conditions specified in the previous example. Substituting the known values for the letters of the formula gives

$$hp = \frac{510 \times 300}{33{,}000} = 4.63$$

The net power thus required is affected also by the efficiency of the machine, which is usually from 60 to 80 per cent, or about 70 per cent on an average. By dividing the net horsepower rating obtained in the last

formula by the efficiency rating, the total, or gross horsepower, is determined. This gross horsepower is expressed by the formula

$$hp_m = \frac{hp}{E}$$

Where

hp_m = gross horsepower necessary
hp = horsepower required to make the cut
E = efficiency of machine

In the preceding problem in which the net horsepower was determined, the motor would have to be

$$\frac{4.63}{.70} = 6.6$$

horsepower capacity in order to handle the load. The closest available size is a seven or seven and one-half horsepower motor.

The pressure on the tool for a given material is independent of the cutting speed, but is dependent on the depth of cut and the feed used. The horsepower, on the other hand, depends on the pressure on the tool and on the cutting speed. It is obvious that the combination of depth of cut, feed per revolution, and the cutting speed should be so adjusted that the demand for power is within the capacity of the motor driving the machine.

There are times when it is desirable to express the performance of a machine in terms of the horsepower required per cubic inch of material removed per minute. The number of cubic inches removed per minute can be found using the formula

$$c = 12 \times f \times d \times S$$

Where

c = cubic inch of metal removed per minute
d = depth of cut in inches
f = advance of tool per revolution in inches
S = cutting speed in feet per minute
12 = constant

These symbols represent the same factors used in previous formulas except c represents cubic inches removed per minute.

Horsepower per cubic inch, then, is

$$\text{hp per cu. in.} = \frac{\frac{P \times S}{33{,}000}}{12 \times f \times d \times S}$$

$$= \frac{P \times S}{33{,}000 \times 12 \times f \times d \times S}$$

$$= \frac{P}{396{,}000 \times f \times d}$$

The application of this formula can be illustrated by assuming it is

desired to find the tangential force or chip pressure on the tool; the horsepower necessary to cut the metal; the horsepower of the motor when the efficiency is 70 per cent; the cubic inches of metal removed per minute; and the horsepower required in terms of cubic inches per minute. A mild steel shaft is being turned at 400 f.p.m. The shaft is 4" in diameter and is being machined with a feed of 1/32" per revolution and a depth of cut of 1/4". Using the formulas discussed previously, the solutions in their respective orders are

$$P = 270{,}000 \times 1/4 \times 1/32 = 2{,}109 \text{ pounds}$$

$$hp = \frac{2{,}109 \times 400}{33{,}000} = 25.6$$

$$hp_m = \frac{25.6}{.70} = 37 \text{ (approximately)}$$

$$\text{Cu. in. per min.} = 12 \times 1/4 \times 1/32 \times 400 = 37.5$$

$$\begin{array}{l}\text{hp per cu. in.} \\ \text{per minute}\end{array} = \frac{25.6}{37.5} = 0.603$$

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Define cutting speed and feed.
2. Define roughing and finishing cuts.
3. A pipe flange is 12" outside diameter, 4" inside diameter, and is being faced with a sintered carbide tool at 100 r.p.m. If the feed is 1/32" per revolution, determine the maximum cutting speed and the time necessary to make the cut.
4. A 3 1/2" diameter 4'0" long shaft is being turned down to 3 1/4" in two cuts. The roughing cut is 3/32" in depth with a feed of 1/32" per revolution. The finishing cut is 1/32" in depth, with a feed of .008" per revolution. The cutting speeds are 250 f.p.m. for roughing and 300 f.p.m. for finishing. Determine the r.p.m. for the roughing cut, the r.p.m. for the finishing cut, the time necessary to take the cut over the entire length, the time necessary to take the finishing cut, and the total time taken for roughing and finishing.
5. Find the pressure on the tool when machining a low alloy steel stud 2" in diameter, at 400 r.p.m., with a feed of 1/64" per revolution and a depth of cut of 1/8".
6. What horsepower is required to drive the machine used under the conditions described in question 5, if the efficiency of the machine is 70 per cent?

7. A cast-iron roll 8" in diameter is turned down to 7 5/8" in one cut. The roll hardness is 200 Brinell. The feed is .020" per revolution. Determine the r.p.m., taking the cutting speed from Table V, and the length of time necessary to cut the 24" length.

8. Determine the tool life when turning S.A.E. 3140 when the depth of cut is 1/8", the feed 1/64" per revolution, the work 4" in diameter, and the turning speed 250 r.p.m. C = 650 and = 0.16.

9. Determine the cubic inches of material removed per minute in question 5. Find the horsepower per cubic inch of material removed per minute.

10. If the feed is changed to 1/32" per revolution and the depth of cut to .1" in question 5, what will be the effect of the chip pressure on the tool?

REFERENCES

Benedict, O., Jr. Manual of Machine Shop Practice. New York: McGraw-Hill Book Co., Inc., 1944.

Boston, O. W. "What Happens When Metal Is Cut," Transactions, A.S.M.E., Vol. 52, No. 11, Paper MSP-52-11, Pp. 119-141, 1930.

Holmes, W. G. "What Should Determine the Depth of Cut?" The National Tool and Die Journal, Vol. 6, No. 6.

McCoy, W. H. "The Future of Tungsten Carbide as a Cutting Tool," Journal S.A.E., Vol. 27, No. 5, p. 557, 1930.

Taylor, F. W. "The Art of Cutting Metals," Transactions, A.S.M.E., Vol. 28, 1907.

Warner & Swasey Co. "Blue Chips," Vol. 3, No. 7.

CHAPTER VII

Carbide Tool Design

Requirements of Carbide Tools. The primary requirements in the design of single-point carbide tools are identical with those involved in the design of high-speed or straight-carbon steel tools. In brief, these tools should be free cutting, have adequate strength, be rigid, and have cutting angles best suited to the job at hand.

Although the application of carbide cutting tools is at present widespread throughout the mechanical industries, a thorough understanding of tool design and construction problems is not universal. Since a complete knowledge of the basic problems confronting the tool designer or engineer will enable him as well as the mechanic to construct and set up the tool that will give the longest and most efficient performance, the following pages have been devoted to the factors which govern the design and construction of carbide tools.

Involved in the design of any cutting tool are these considerations:

1. The material to be cut — its physical and metallurgical properties.
2. The depth of cut desired and the feed at which the machine is to be operated.
3. The rate at which the material is to be removed. This point is of prime importance, for if the speed wanted is not high enough, nothing can be gained by substituting carbide tools for high-speed steel.
4. The power available at the machine.
5. The rigidity of the machine.
6. The rigidity of the work itself.
7. The nature of the operation, such as turning, facing, boring, planing, shaping, or other.
8. Is the cut continuous or interrupted.
9. Is the machine capable of operation at speeds high enough to warrant the use of carbide without undue stress and wear?
10. The grade of carbide to be used.
11. The methods of brazing available.
12. The methods of grinding available.
13. The forces acting on the tool.
14. The tool angles that may be most suitable for cutting the material at hand.

In addition, design involves the selection of the proper size of tool

blank and of the shank, the specification of steel for the shank, the heat treatment (when necessary), and the type of finish required on the tool.

Forces Acting upon the Tool. The resistance of the metal to the cutting action of the tool produces forces which the designer must take into consideration. Experienced designers and tool makers would seem to have a sort of knack or insight in choosing the right size of tool for a particular job. Actually, their selection is a matter of sound judgment, based on long years of experience. Occasionally, however, this rule-of-thumb method of selection results in costly failures. A good tool designer will consider first, the cutting forces which will be in operation on the tool, and will then alter the design or select a proper size tool to fit the particular situation before the failure occurs.

The cutting forces acting on the tool were enumerated in Chapter VI as tangential, radial, and axial. The formula for determining the tangential force exerted on the tool also was given in Chapter VI and may be used here for computation purposes. Values for constants for different materials may be obtained from Table X of the same chapter. These values are approximate but may be used safely for estimating and designing purposes. As a means of refreshing the memory and making certain that the use and importance of the chip pressure formula is thoroughly understood, assume it is expected that a turning tool will be used on a general run of jobs in a shop where soft and alloy steels are to be machined. At times it may be necessary to take a 3/16'' cut with a feed of 1/64''. What pressure may be expected at the tip of the tool? The material constant (K) is taken from Table X. Substituting values for the letters of the formula gives

$$P = K \times f \times d$$

$$P = 375{,}000 \times 1/64 \times 3/16$$

$$P = 375{,}000 \times .0156 \times .1875$$

$$P = 1{,}097 \text{ pounds}$$

Power Needed for Cutting. For successful performance, carbide tools require higher speeds than do the high-speed steel tools. Higher speeds call for more power. It is important, therefore, that the power formulas discussed in Chapter VI be remembered. As a refresher, assume that it is desired to know what horsepower motor will be required to operate the machine in the preceding problem if the cutting speed is to be 200 f.p.m. and the efficiency of the machine is rated at 75 per cent. Substituting values for the letters in the formulas gives

$$hp = \frac{P \times S}{33{,}000}$$

$$hp = \frac{1,097 \times 200}{33,000}$$

$$hp = 6.65$$

which is the net horsepower required for the machining operation. The gross horsepower is expressed by the formula

$$hp_m = \frac{hp}{E}$$

$$hp_m = \frac{6.65}{.75}$$

$$hp_m = 8.8 \text{ or } 9, \text{ approximately.}$$

To meet these conditions as well as recognize the requirements of motor design, the horsepower of the machine should be about ten for continuous operation. Since electric motors are capable of being over-

TABLE I. RADII IN INCHES OF STANDARD TOOL BLANKS

W	R
1/8 to 1/4	1/8
9/32 to 3/8	3/16
3/8 and over	1/4

loaded to some extent, a smaller motor, one probably of about seven-and-a-half horsepower, would be satisfactory in a shop where cutting is not done over long periods of time.

Rigidity of Machine. It has been carefully pointed out previously that for successful carbide application, the machine must be rigid and free from any vibration. This means that spindle bearings should be adequate and well fitted. All sliding parts should be adjusted so they are free from loose play. These conditions are absolutely necessary. There is nothing more deteriorating to carbide tool cutting edges than vibration of the work. Under such conditions, the carbide edges chip, causing the tool to fail prematurely.

Rigidity of the Tool. Another important prerequisite to the successful use of carbide tools is the rigidity of the tool shank and the tool holding device. These points will be discussed more fully later in the chapter. Even when the machine and the work are rigid, a nonrigid tool shank or a poor tool holder will contribute to early tool failure. These are all important points which the tool designer cannot overlook.

Blanks for Carbide Tools. Manufacturers of carbide tools and blanks supply both standard and special blanks. Standard blanks can be purchased from manufacturers' stocks while special blanks are

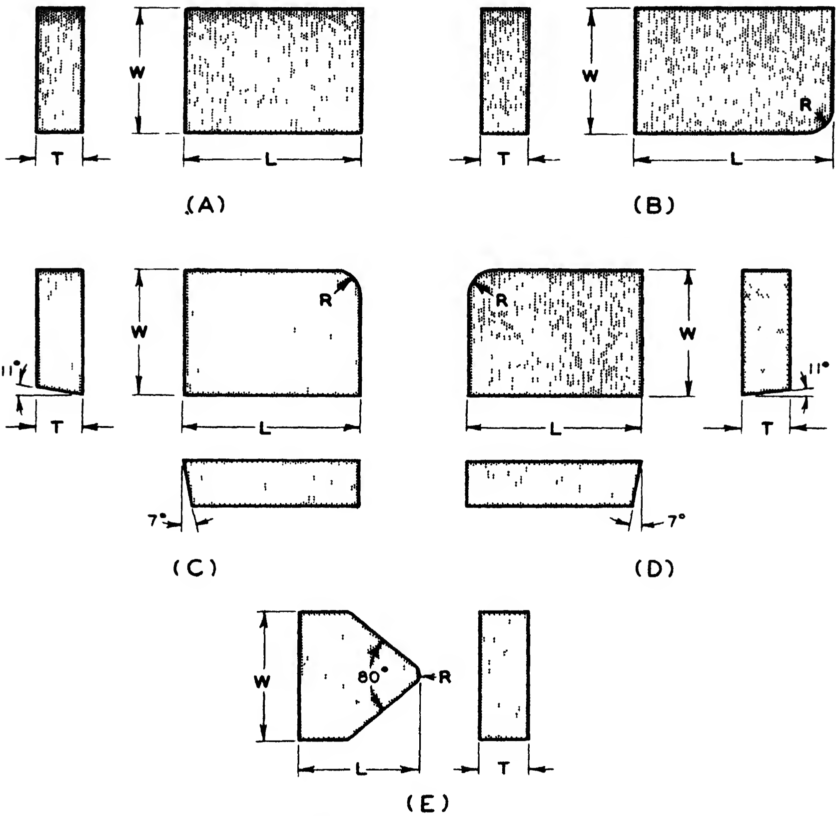


Fig. 1. Representative Standard Blanks in Use Throughout Industry.
(A) Rectangular Blanks, (B) Blanks with Radius on One Corner, (C) Right-Hand Standard Blank, (D) Left-Hand Standard Blank, (E) Roundnose Tool Blank

usually made to order. This is an important factor to be considered by the tool designer since standard blanks can be purchased more economically than the special blanks and are obtained more easily and quickly. Representative standard blanks in use throughout the industry are shown in (A) through (E) inclusive of Fig. 1.

At (A) in Fig. 1 is pictured a rectangular blank which finds wide application in both single- and multiple-point cutting tools. The blanks are made sufficiently oversize to permit grinding on the sides, on top and bottom, and on both ends.

In (B) is illustrated a popular blank which has a radius on one corner. This blank is used when the tool has an end-milled recess in the shank. These blanks are the same as those pictured in (A) except for the radius which is standardized. Details of this radius are given in Table I.

TABLE II. SIZES IN INCHES OF STANDARD TOOL BLANKS

Dimensions			Dimensions		
Thickness (T)	Width (W)	Length (L)	Thickness (T)	Width (W)	Length (L)
1/16	1/8	5/8	3/16	3/8	3/4
	3/16	1/4		7/16	5/8
	1/4	5/16		7/16	13/16
				1/2	1/2
3/32	3/16	5/16		1/2	3/4
	3/16	1/2		3/4	3/4
	1/4	3/8			
	1/4	1/2	1/4	3/8	9/16
	5/16	3/8		3/8	3/4
	3/8	3/8		7/16	5/8
	3/8	1/2		1/2	3/4
				9/16	1
1/8	3/16	3/4		5/8	5/8
	1/4	1/2		3/4	1
	1/4	5/8			
	1/4	3/4	5/16	7/16	5/8
	5/16	7/16		7/16	15/16
	5/16	1/2		1/2	3/4
	5/16	5/8		1/2	1
	3/8	1/2		5/8	1
	3/8	3/4		3/4	3/4
	1/2	1/2		3/4	1 1/4
	1/2	3/4			
			3/8	1/2	3/4
5/32	3/8	9/16		1/2	1
	3/8	3/4		5/8	1
	5/8	5/8		3/4	1 1/4
				3/4	1 1/2
3/16	5/16	7/16			
	5/16	5/8	1/2	3/4	1
	3/8	1/2		3/4	1 1/4
	3/8	5/8		3/4	1 1/2

A right-hand standard blank is shown in (C) of Fig. 1. Note that relief angles are formed when the blank is manufactured. This procedure saves grinding. However, there is enough material allowed on the sides and length to permit finishing by grinding. The radii of these blanks are the same as given in Table I.

A left-hand standard tool blank is seen in (D) of Fig. 1. This blank also has preformed side and end relief angles. Relief angles are usually provided on blanks with thicknesses greater than 3/16". The popular

TABLE III. RADII IN INCHES OF ROUNDNOSE TOOL BLANKS

T	W	L	R
3/32	5/16	3/8	1/32
3/32	3/8	1/2	1/16
1/8	1/2	9/16	1/16
5/32	5/8	5/8	3/32
3/16	3/4	3/4	3/32
1/4	3/4	1	1/8

styles presented in these four figures are normally carried in stock by manufacturers in the sizes shown in Table II.

A standard blank used for making a roundnosed tool which is employed for light turning, chamfering, or grooving operations, is shown

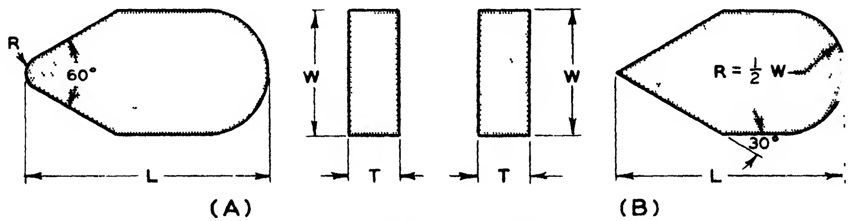


Fig. 2. Additional Blanks Considered More or Less Standard in Industry

in (E) of Fig. 1. The radius R varies with the size of the blank as shown in Table III.

Other more or less standard tool blanks are shown in (A) and (B) of Fig. 2. These blanks are also carried in stock frequently by the manufacturer or his outlets. Such blanks are usually available in the sizes shown in Table IV.

TABLE IV. SIZES IN INCHES OF SEMISTANDARD TOOL BLANKS

Style (A), Fig. 2				Style (B), Fig. 2		
T	W	L	R	T	W	L
1/8	3/4	3/4	1/16	3/32	3/16	7/16
1/8	1/4	9/16	1/16	3/32	3/16	1/2
5/32	5/16	5/8	3/32	1/8	1/4	5/8
3/16	3/8	3/4	3/32	5/32	5/16	3/4
1/4	1/2	1	1/8	1/4	3/8	7/8

Thickness of the Blanks. The thickness of the tool blank depends on the depth of cut taken, the feed of the tool per revolution, and,

TABLE V. THICKNESS IN INCHES OF TOOL BLANKS FOR SINGLE POINT TOOLS BASED ON DEPTH OF CUT AND FEED PER REVOLUTION*

Feed in Inches per Rev.	Depth of Cut in Inches												
	1/64	1/16	3/32	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	7/8	1
.004	1/16	3/32	1/8	1/8	1/8	1/8	1/8	1/8	1/8	5/32	5/32	5/32	1/4
.008	3/32	1/8	1/8	5/32	5/32	5/32	5/32	3/16	3/16	3/16	1/4	1/4	1/4
.012	1/8	1/8	1/8	5/32	5/32	5/32	3/16	3/16	1/4	1/4	5/16	5/16	5/16
.016	1/8	5/32	5/32	5/32	3/16	1/4	1/4	1/4	5/16	5/16	5/16	3/8	3/8
.020	1/8	5/32	5/32	3/16	1/4	1/4	1/4	5/16	5/16	3/8	3/8	3/8	3/8
.024	5/32	5/32	3/16	3/16	1/4	1/4	5/16	5/16	5/16	3/8	3/8	3/8	1/2
.028	5/32	3/16	3/16	1/4	1/4	5/16	5/16	3/8	3/8	3/8	3/8	1/2	1/2
.032	5/32	3/16	1/4	1/4	5/16	5/16	3/8	3/8	3/8	3/8	1/2	1/2	1/2
.036	3/16	1/4	1/4	1/4	5/16	5/16	3/8	3/8	3/8	1/2	1/2	1/2	1/2
.040	3/16	1/4	1/4	5/16	5/16	5/16	3/8	3/8	3/8	1/2	1/2	1/2	1/2
.044	3/16	1/4	1/4	5/16	5/16	5/16	3/8	3/8	3/8	1/2	1/2	1/2	1/2
.048	3/16	1/4	1/4	5/16	5/16	3/8	3/8	3/8	1/2	1/2	1/2	1/2	1/2
.052	1/4	1/4	1/4	5/16	5/16	3/8	3/8	3/8	1/2	1/2	1/2	1/2	1/2
.054	1/4	1/4	1/4	5/16	5/16	3/8	3/8	1/2	1/2	1/2	1/2	1/2	1/2
.058	1/4	1/4	5/16	5/16	3/8	3/8	1/2	1/2	1/2	1/2	1/2	1/2	1/2
.062	1/4	1/4	5/16	5/16	3/8	3/8	1/2	1/2	1/2	1/2	1/2	1/2	1/2

*For interrupted cuts, select one size larger up to 1/2".

Courtesy of Vascoloy-Ramet Corp

most important, on the nature of the material cut. Experience has shown that tool blanks listed in Table V are adaptable to the cutting of alloy steel with single-point tools. When a material is of such a nature that excessively heavy forces are not exerted on the tool, a thinner tool blank may be used for the tool tip. For cutting extremely tough, hard,

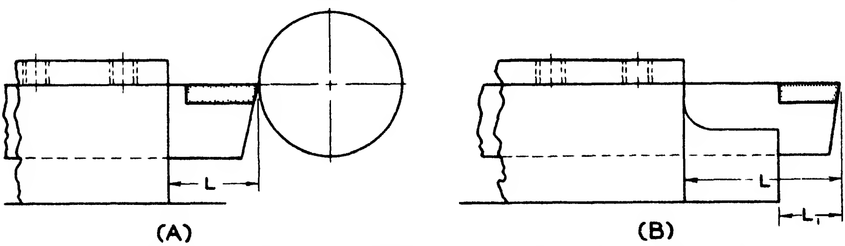


Fig. 3. Conditions of Overhang That Will Bring About Varying Deflection in the Tool

and strong materials, and for taking interrupted cuts, tool blanks should be thicker than those specified in Table V.

Shanks for Carbide Tools. The shank size is usually governed by the tool holding device on the machine, the depth of cut to be taken, the material to be cut and its properties, and the overhang which

may be necessary to perform a particular operation. In (A) and (B) of Fig. 3 are illustrated two conditions of overhang which will result in different amounts of deflection of the tool. Where such conditions prevail, the shank in (B) would have to be much heavier than that shown at (A), or the tool holder would have to be modified so as to give more support to the shank.

Standard Sizes of Shanks. The standard sizes of shanks made by the manufacturers of sintered carbide tools are given in Table VI. Shanks are either square or rectangular in cross section. Other shanks

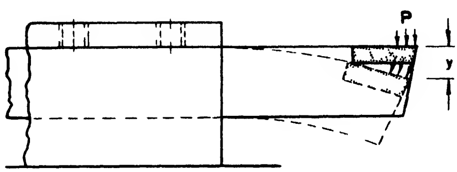


Fig. 4. Exaggerated Drawing Showing How the Cutting Force Causes Deflection

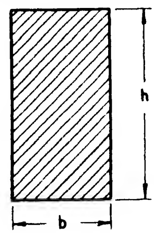


Fig. 5. Cross Section of a Rectangular Shank

which are made to meet special requirements may be semistandard or special.

Strength and Deflection. When the cutting force is applied to the tool, as when a cut is taken, the point of the tool will deflect somewhat. An exaggerated drawing of this is given as Fig. 4 in which P is the cutting force or tangential force, and y is the deflection caused. The

TABLE VI. DIMENSIONS IN INCHES OF STANDARD SHANKS

Square		Rectangular	
1/4	3/4	1/2 × 3/4	3/4 × 1 1/2
5/16	7/8	1/2 × 1	3/4 × 2
3/8	1	5/8 × 1	1 × 1 1/4
1/2	1 1/4	5/8 × 1 1/4	1 × 1 1/2
5/8	1 1/2	3/4 × 1	1 1/2 × 2

forces acting on a tool were discussed in Chapter VI. From this it will be recalled that the tangential force was expressed by the formula $P = K \times f \times d$.

This force, or pressure, on the tool which causes deflection, creates stress in the shank. The stress for this type of tool is greatest at the support, and the deflection is most pronounced at the point of contact of the tool with the work. Under these conditions, the tool may be thought of as a cantilever beam. The stress at the support is the result of the bending action resulting from force P. For practical purposes,

force P may be considered as being concentrated at the point of the tool. The bending moment at the support is expressed by the formula

$$M_b = P \times L$$

M_b = the bending moment in inch pounds
P = the pressure on the tool in pounds
L = the distance from the application of force P to the support, in inches

If the force on a tool were found to be 1,100 pounds and the distance from the tool tip to the tool support (L) were 1 1/2", the bending moment would be 1,650 inch pounds. The solution consists merely of substituting the known values for the letters in the formula.

The bending moment is resisted by the material because of its strength. Therefore, there is a resisting moment in the material which, for conditions of equilibrium, must be equal to the bending moment. The resisting moment, then, is expressed by the formula.

$$R = \frac{S \times b \times h^2}{6}$$

R = the resisting moment
S = the stress in pounds per square inch
b = the width or thickness of the shank in inches, as shown in Fig. 5
h = the height of the shank in inches, as shown in Fig. 5
6 = a constant

Equating these two formulas results in

$$P \times L = \frac{S \times b \times h^2}{6}$$

By transposing the terms in this formula, it is possible to solve for any one of the values by means of these formulas

$$L = \frac{S \times b \times h^2}{6 \times P}$$

$$S = \frac{6 \times P \times L}{b \times h^2}$$

$$B = \frac{6 \times P \times L}{S \times h^2}$$

$$b = \sqrt{\frac{6 \times P \times L}{S \times b}}$$

It is apparent that in a square shank, b will be equal to h (Fig. 5).

Therefore, b may be substituted for h, giving

$$P \times L = \frac{S \times b^3}{6}$$

Assuming that the tool shank in the last problem was 3/4" in thickness and that h was 1", the stress at the support, all other conditions remaining the same, is 13,200 pounds per square inch. The solution is obtained by substituting the known values for the letters in the formula, resulting in

$$S = \frac{6 \times 1{,}100 \times 1.5}{3/4 \times 1 \times 1} = 13{,}200 \text{ p.s.i.}$$

If L had been 1 1/2" high instead of 1" as in the problem just solved, the stress on the shank, other conditions remaining the same, would be

$$S = \frac{6 \times 1{,}100 \times 1.5}{3/4 \times 1.5 \times 1.5} = 5{,}892 \text{ p.s.i.}$$

Permissible Shank Stress. Permissible stresses, commonly known as design stresses, are usually low in carbide tools. Tool shanks are generally made of S.A.E. 1045, 1060, 1085, or 1095 steels. Alloy steels may be used when more strength is required. The hardness of the shank after brazing varies, and may range from 20 to 30 on the Rockwell C scale. This results in a tensile strength of tool shanks of about 70,000 to 90,000 p.s.i.

These values represent the ultimate strength of the material and are not to be used in full for design purposes. A much smaller stress value, known as "design stress," is used by most good designers. This practice tends to make the shank much safer than it actually needs to be from the standpoint of stress. Design stress is obtained by dividing the ultimate stress by a number called the "factor of safety." For tools of this type, the safety factor number may range from 10 to 20, providing for all manner of unforeseen operating contingencies.

Thus, with a safety factor of 10 and an ultimate stress of 70,000, the design stress would be 7,000 p.s.i. With a safety factor of 20, the design stress would be 3,500 p.s.i. The designer, when he knows the operating conditions, will use the proper design-stress values to arrive at the correct shank dimensions. Also, the tool engineer, when he selects a shank, can compute the stress and determine whether it falls near the desired points.

To cite a problem in tool design, assume it is desired to determine the height of a rectangular tool shank when force P is 850 pounds; the thickness of the shank, or dimension b, is 1"; the overhang (L) is 1 1/2"; and the permissible stress is not to exceed 5,000 p.s.i. The formula used is the last of those obtained by transposing the terms of the formula given for finding the resisting moment. Substituting values

for the transposed terms in the formula results in

$$h = \sqrt{\frac{6 \times 850 \times 1.5}{5000 \times 1}} = \sqrt{1.53} = 1.24 \text{ inches}$$

This is approximately 1 1/4'', the height necessary for a tool shank 1'' thick which must resist a tangential pressure of 850 pounds at 1 1/2'' of overhang.

Design for Shank Deflection. While shanks designed on the basis of strength alone may be satisfactory, it is not often that they are produced in this manner. It is easily possible to select a shank strong enough for the job, yet have it deflect under the cut to the extent that it would be the source of chatter. This would result in a poor finish on the work and early tip failure—a point frequently overlooked by some designers and mechanics. The most important factor in good, single-point tool design is the rigidity of the shank. The shank must be capable of resisting deflection under its cutting load.

In dealing with this problem, the shank may be considered as a cantilever beam as before, loaded at one end as shown in Fig. 4. The maximum deflection is expressed by a formula which is well known to mechanical engineers.

$$y\ (\text{max.}) = \frac{P \times L^3}{3 \times E \times I}$$

y = maximum deflection in inches
P = pressure on the tool in pounds
L = the distance from the application of force P to the support, in inches
E = modulus of elasticity of the shank material which, for steel, is taken at 30,000,000
I = the moment of inertia of the cross-section area of the shank shown in Fig. 5, expressed by the formula

$$I = \frac{b \times h^3}{12}$$

Substituting the formula for the moment of inertia in the formula for determining maximum deflection results in

$$y = \frac{P \times L^3}{3 \times E \times \frac{b \times h^3}{12}}$$

$$y = \frac{4 \times P \times L^3}{E \times b \times h^3}$$

If it is assumed that some deflection is permissible, either b or h can be solved for under any condition of P and L. The terms are merely substituted in the formula, resulting in

$$h = \sqrt[3]{\frac{4 \times P \times L^3}{E \times b \times y}}$$

or

$$b = \frac{4 \times P \times L^3}{E \times h^3 \times y}$$

In order to understand more clearly the application of these formulas relating to shank deflection, assume it is desired to determine the height of a steel shank which is 3/4'' wide. There is an overhang of 1 1/2'', and the cutting force of 850 pounds should not cause more than a .001'' deflection. The known values are substituted for the letters in the formula, resulting in

$$h = \sqrt[3]{\frac{4 \times 850 \times 1.5 \times 1.5 \times 1.5}{30{,}000{,}000 \times 0.75 \times .001}}$$

$$h = \sqrt[3]{.510}$$

$$h = .799''$$

If the overhang in the last problem were increased to 2.25'', and the permissible deflection to .002'', the height of the steel shank, other conditions remaining the same, should be

$$h = \sqrt[3]{\frac{4 \times 850 \times 2.5 \times 2.5 \times 2.5}{30{,}000{,}000 \times 0.75 \times 0.002}}$$

$$h = \sqrt[3]{1.180}$$

$$h = 1.056''$$

If the shank in the problem just worked were specified to be 3/4'' × 1 1/4'', and all other conditions remained the same, the stress to which it would be subjected during cutting action would be

$$S = \frac{6 \times P \times L}{b \times h^2}$$

$$S = \frac{6 \times 850 \times 2.5}{0.75 \times 1.25 \times 1.25}$$

$$S = 10{,}897 \text{ p.s.i.}$$

If a 1 1/2" square shank were selected for a tool having an overhang of 3", the deflection of the tool would vary with the cut taken. Assuming the cut to be of 3/8" depth at 1/32" feed in mild steel, the deflection would be

$$y\ (\text{max.}) = \frac{P \times L^3}{3 \times E \times I}$$

Force P is found by using the formula given in Chapter VI.

$$P = K \times f \times d$$

Substituting values for letters (K is taken from Table X, Chapter VI), gives

$$P = 270{,}000 \times .0312 \times .375$$

$$P = 3{,}159 \text{ pounds}$$

$$y = \frac{4 \times 3{,}159 \times 3^3}{30{,}000{,}000 \times 1.5 \times 1.5^3}$$

$$y = .002''$$

It is interesting to note that the deflection of the shank increases or decreases directly as the cube of the overhang, L. Therefore, if the overhang in the last problem just solved should be reduced to 2", the deflection would be decreased and could be obtained by multiplying the value of y by the ratio of the two overhangs cubed.

$$y = .002'' \times \frac{2^3}{3^3} = .0005$$

The same value will be obtained by substituting 2 for 3 in the formula used for determining maximum deflection.

Deflection of the tool cannot be reduced by the substitution of alloy steel for carbon steel, or by hardening the shank. The deflection is a part of the elastic property of the material from which the shank is made. Thus, it does not change much as the result of additions of alloying elements to the steel, or by hardening or tempering. This is one of the most common errors made by designers and mechanics. Too many of these people try to increase the rigidity of the tool through heat treatment, or by the selection of more expensive material for the shank.

Deflection and stress are two of the most troublesome elements of tool design, carbide or otherwise. Any appreciable chatter or lack of rigidity is the surest way to ruin a carbide tool, thus making the job more costly instead of more efficient.

Standard Tools. The tools shown in Fig. 6 comprise the styles most commonly used in turning, facing, boring, grooving, and chamfering operations. Some of these tools are stock items. Others are made to

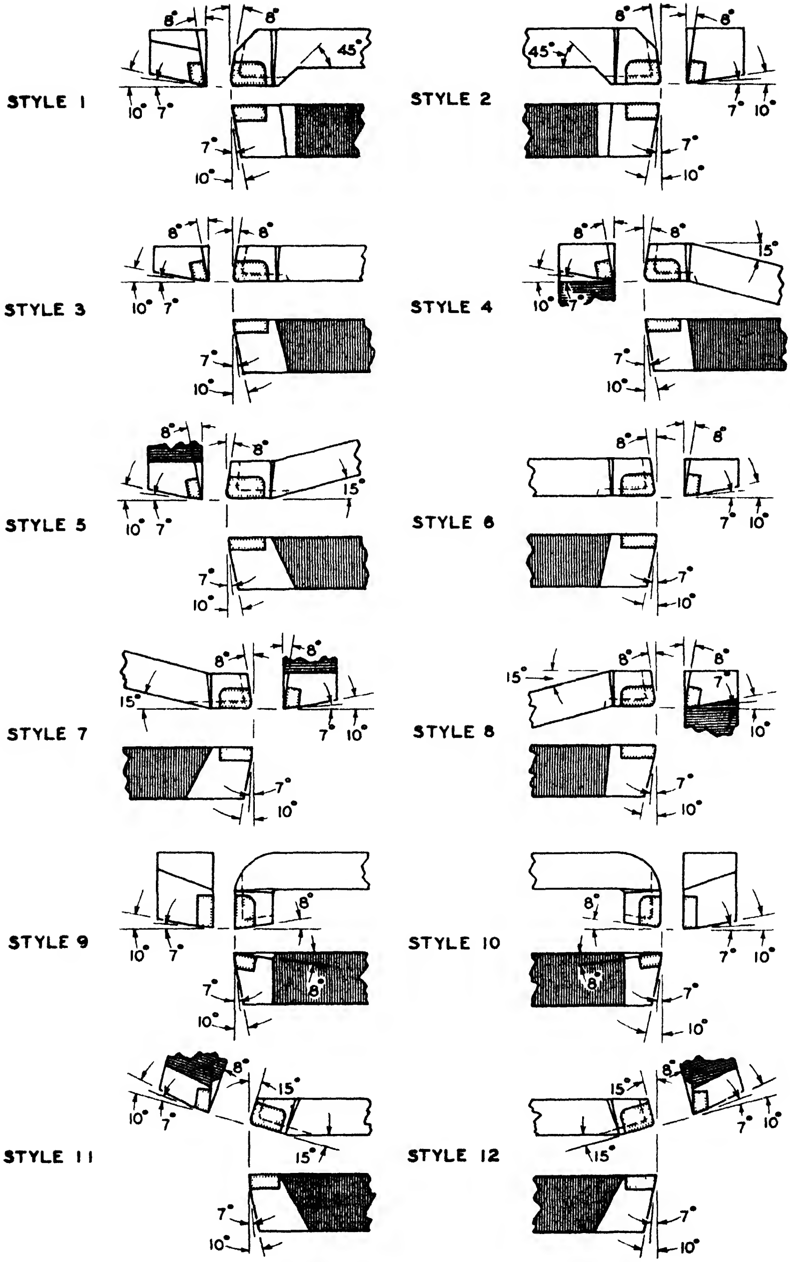


Fig. 6. Standard, Single-Point Carbide Tools

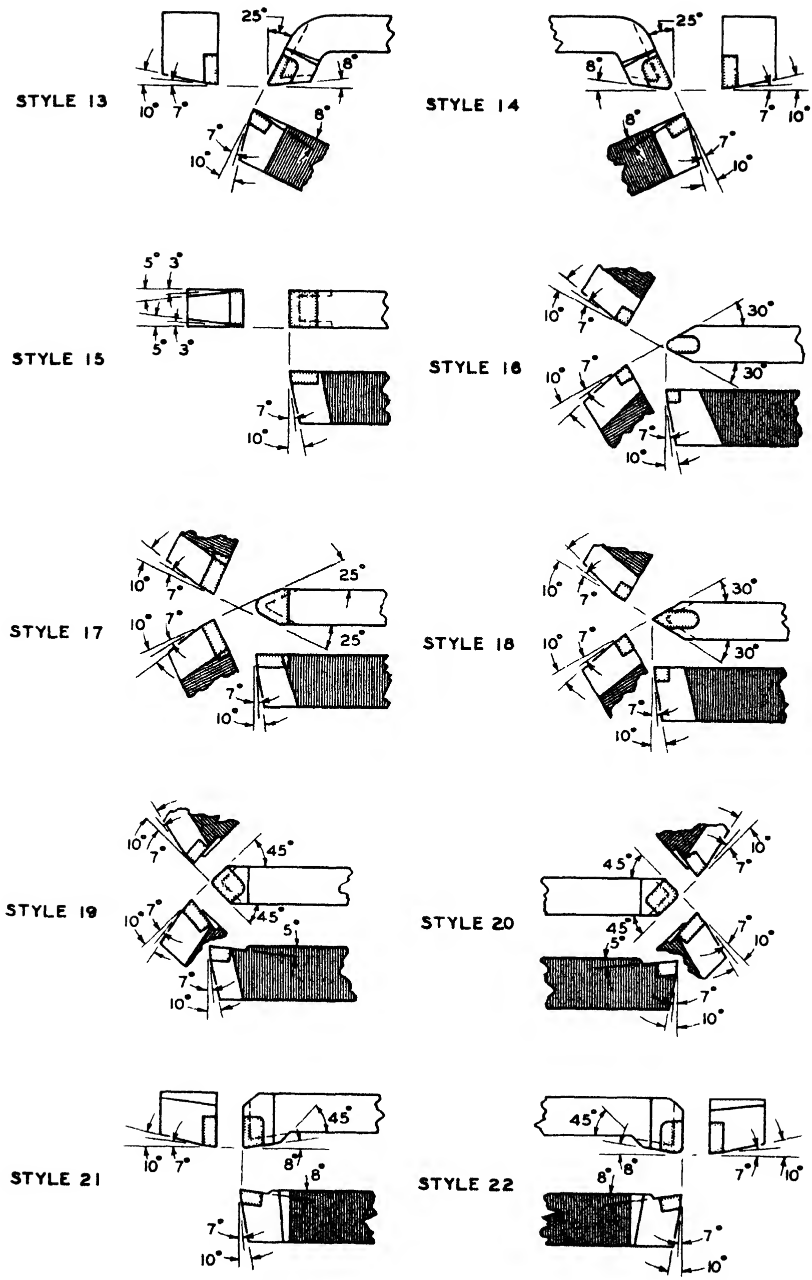


Fig. 6 *(Continued)*. Standard Single-Point Carbide Tools

Courtesy of Vascoloy-Ramet Corp

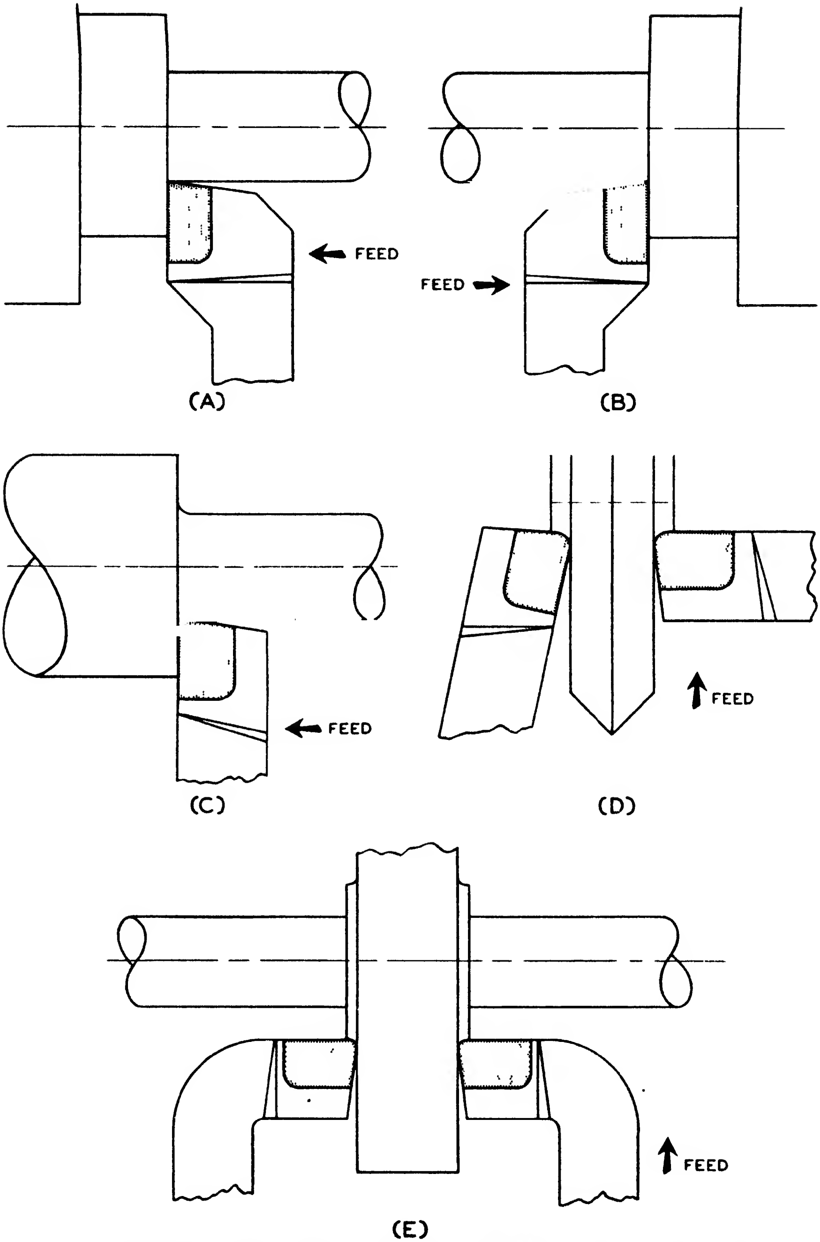


Fig. 7. Examples of the Applications of the Various Designs of Carbide Tipped Shanks. (A) Use of the Right-Hand Offset Tool for Turning, (B) Use of the Left-Hand Offset Tool for Turning, (C) Turning to a Square Shoulder, (D) Facing to a Square Shoulder, (E) Bent-Shank Tool Setup for Straddle Facing

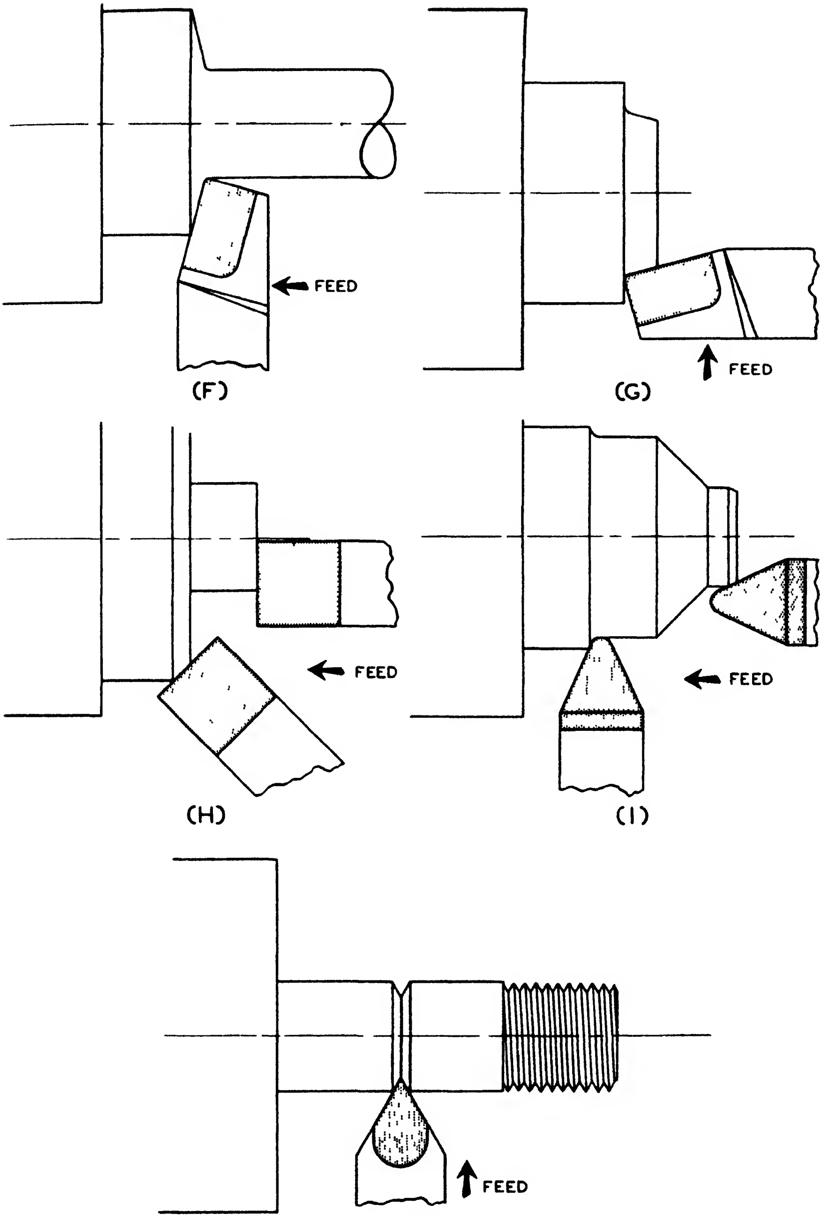


Fig. 7 ***(Continued)*. Examples of the Applications of the Various Designs of Carbide Tipped Shanks.**
(F) Tool Used for Straight Turning, (G) Tool Used for Facing or Chamfering Work, (H) Setup for Square-Nosed Tools, (I) Turning and Chamfering Tools, (J) Threading or V Grooving Tool

order in any desired quantity and are tipped with a suitable grade of carbide.

Style 1 is a right-hand tool which is used for turning against a shoulder or close to a chuck as shown in (A) of Fig. 7. Style 2 in Fig. 6 is used for feeding from left to right against a shoulder as shown in

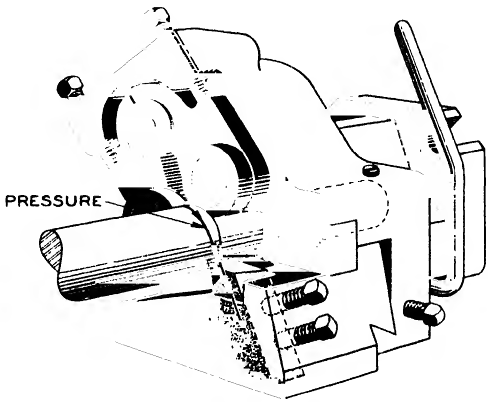


Fig. 8. Application of the Roller Turner Tool

(B) of Fig. 7. Styles 3 and 6 are right- and left-hand straight turning tools which are used for turning to a square shoulder and for boring against a square shoulder in a large bore. This tool is shown in (C) of Fig. 7. The left-hand tool can be used for facing to a shoulder as shown in (D) of Fig. 7.

Styles 4 and 5, right-hand tools with shanks bent to the right or left, are used in boring mills and lathe work for setups where the tool has to reach over for the cut. Styles 7 and 8 are left-hand boring, turning, or facing tools which may be used singly or in pairs for facing operations, These tools have slightly beveled edges to give them better performance.

Styles 9 and 10 are right- and left-hand, bent-shank, facing tools which may be used singly or in pairs for facing operations. An example of their use in pairs is shown in (E) of Fig. 7. Style 9 is frequently used in a vertical boring mill for boring against a shoulder in large holes. Styles 11 and 12 are tools with a side cutting-edge angle or a lead angle. They are commonly used in lathe or boring mill work. Two applications of these tools are shown in (F) and (G) of Fig. 7.

Styles 13 and 14 are tools that are used for facing and boring operations either singly or in a setup where either or both tools may be brought into action. Style 15 is a broadnosed tool used for turning, facing, and chamfering operations, sometimes in a setup such as is shown in (H) of Fig. 7. Style 16 is used for grooving and profile cutting. Style 17 is used for light turning cuts, as shown in (I) of Fig. 7, and for chamfering. Style 18 is a threading tool or may be used for V grooving as shown in (J) of Fig. 7.

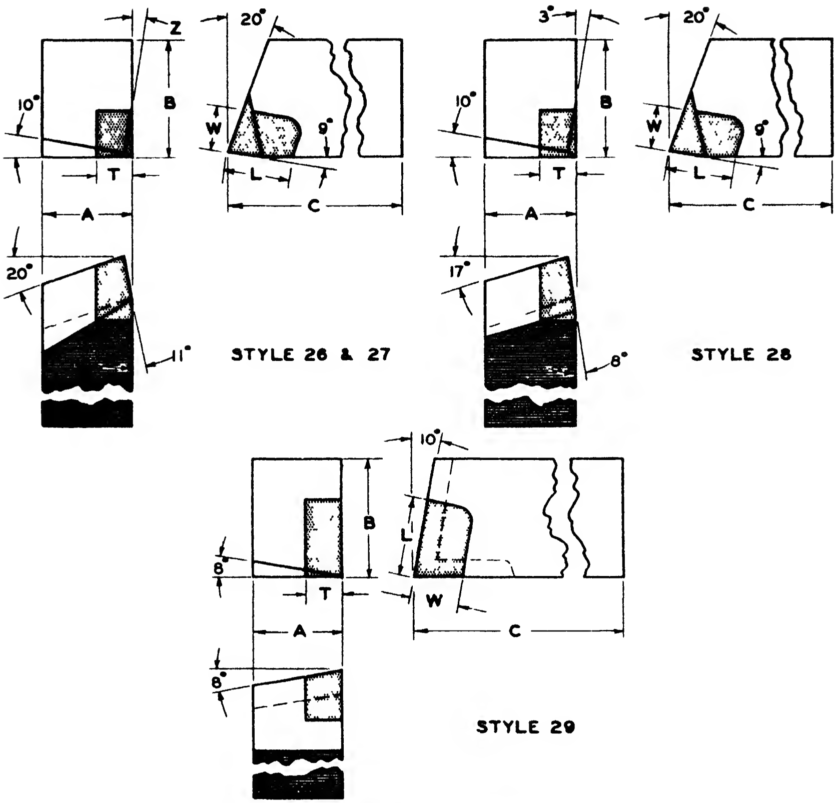


Fig. 9. Various Types of Roller Turner Cutting Tools

Styles 19 and 20 are tools with large lead angles and are used for turning, facing, boring, and chamfering, usually on lighter cuts. Styles 21 and 22 and left- and right-hand tools which can be used singly or in a combination for straddle facing of flanged work as was shown in (E) of Fig. 7.

The tools stocked by most manufacturers are styles 1, 2, 3, 6, 11, 12, 15, and 17. They are obtainable with shanks as listed in Table VI, with the exception of styles 15 and 17, which are listed in Table VII. Letters A, B, and C in the head of Table VII refer to the over-all dimensions of the tool, whereas the letters T, W, and L refer to the dimensions of the carbide tip only. These dimensions are illustrated in Fig. 9.

Other tools which are commonly carried in stock are the roller turner tools of the type illustrated in Fig. 8. These tools are used in connection with turret lathe work. The individual tools themselves are shown in Fig. 9. Their dimensions are given in Table VIII. The letters in the table head refer to the shank and blank dimensions.

TABLE VII. DIMENSIONS OF STYLES 15 AND 17 STOCK TOOLS

Style 15

A	B	C	T	W	L	R
1/4	1/4	1 1/2	1/16	1/4	5/16	-
5/16	5/16	2 1/4	3/32	5/16	3/8	-
3/8	3/8	2 1/2	3/32	3/8	3/8	-
1/2	1/2	3 1/2	1/8	1/2	1/2	-
5/8	5/8	4	5/32	5/8	5/8	-
3/4	3/4	4 1/2	3/16	3/4	3/4	-
1	1	7	1/4	1	3/4	-
1 1/4	1 1/4	8	5/16	1 1/4	3/4	-
5/8	1	6	1/4	5/8	5/8	-
1/2	1	6	3/16	1/2	1/2	-
5/8	1 1/4	7	1/4	5/8	5/8	-
3/4	1	6	1/4	3/4	1/2	-
3/4	1 1/2	8	5/16	3/4	3/4	-
1	1 1/2	8	5/16	1	5/8	-

Style 17

A	B	C	T	W	L	R
5/16	5/16	2 1/4	3/32	5/16	3/8	1/32
3/8	3/8	2 1/2	3/32	3/8	1/2	1/16
1/2	1/2	3 1/2	1/8	1/2	9/16	1/16
5/8	5/8	4	5/32	5/8	5/8	3/32
3/4	3/4	4 1/2	3/16	3/4	3/4	3/32

TABLE VIII. DIMENSIONS OF ROLLER TURNER TOOLS

	A	B	C	T	W	L
For Warner & Swasey Turret Lathes	3/8	1/2	1 1/2	3/16	1/4	3/8
	1/2	3/4	2 5/8	1/4	7/16	9/16
	3/4	1	3	5/16	9/16	11/16
	7/8	1 1/8	3 5/8	5/16	5/8	3/4
	1	1 1/4	4	5/16	5/8	3/4
For Gisholt Turret Lathes	1	1	3 1/2	5/16	9/16	11/16
	1 1/4	1 1/4	4	3/8	5/8	3/4
For Jones & Lamson Turret Lathes	3/8	1/2	2 1/2	3/16	1/4	3/8
	3/4	3/4	3 1/4	1/4	7/16	9/16
	3/4	1	4 1/4	5/16	9/16	11/16
	1	1 1/4	5 1/4	5/16	5/8	3/4

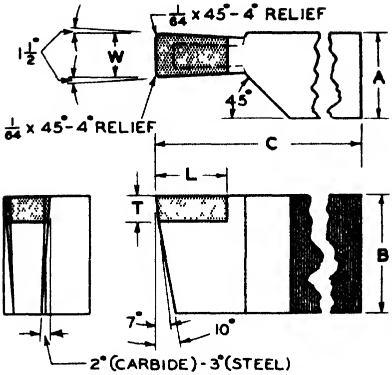


Fig. 10. Standard Design for Cutoff Tools

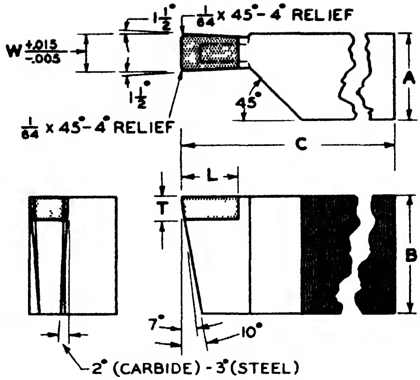


Fig. 11. Standard Design for Grooving Tools

Cutoff tools are shown in Fig. 10. They are also usually stock items. Cutoff tools, as the name suggests, are used for cutting off hollow pieces of work such as shells and pipes. The standardized sizes are given in Table IX.

TABLE IX. SIZES OF CUTOFF TOOLS

A	B	C	T	W	L
5/8	1 1/4	6	3/16	3/8	1/2
3/4	1 1/2	6	1/4	7/16	5/8
3/4	2	7	1/4	7/16	5/8
1	1 1/2	7	1/4	7/16	5/8
1	2	8	1/4	1/2	3/4

Grooving tools, shown in Fig. 11, are used for cutting external grooves or slots, and internal grooves in large holes. Dimensions for this type of tool are given in Table X. These are usually narrower than the cutoff tools.

TABLE X. DIMENSIONS FOR GROOVING TOOLS

W	T	L
Over .060	3/32	1/2
through	3/32	1/2
.200	3/32	1/2
.201	1/8	1/2
through	1/8	1/2
.330	1/8	1/2

Shear-Type Tools. For planing, shaping, and for taking interrupted cuts in a boring mill or a lathe, the shear-type tool is used. Such a tool is illustrated in (A) of Fig. 12. Table XI gives dimensions for standard tips and shanks. It should be noted that the shear-type

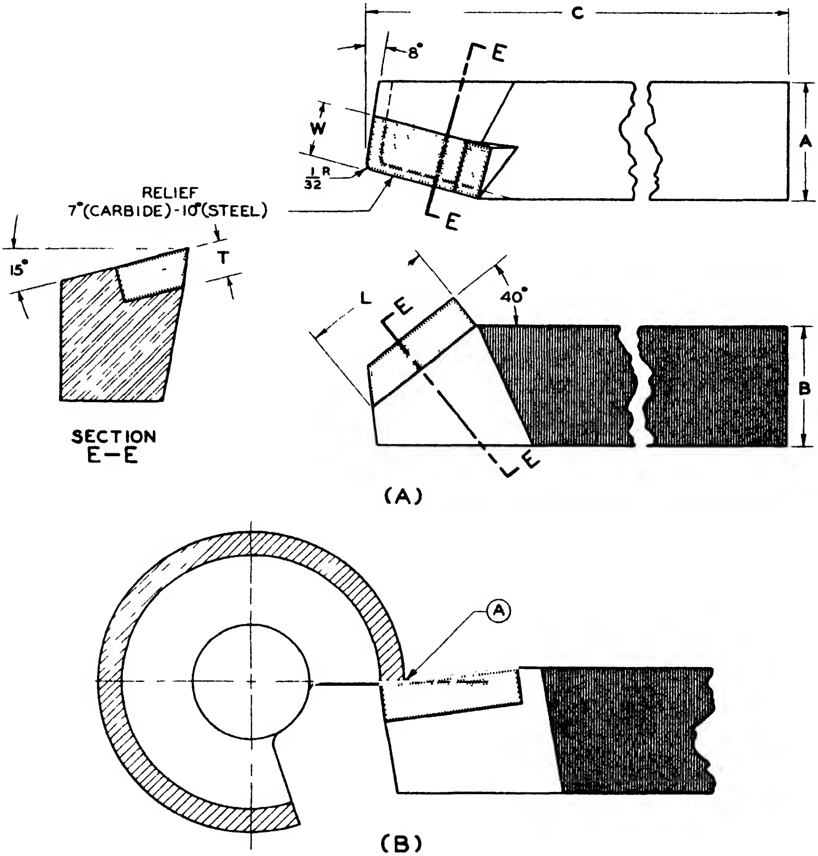


Fig. 12. The Shear-Type Tool (A), and How the Shear-Type Tool Works in an Interrupted Cut (B)

turning tool is of a heavier construction than the regular turning tool. Its resistance to tip breakage is attributed to the fact that it starts cutting away from the point of the tool where the tip is more capable of absorbing the impact of the work. This is shown in (B) of Fig. 12 where the work-tool impact is indicated at A. It should be noted that the negative back rake angle is less in (B).

It is of utmost importance that the tool have negative back rake whenever interrupted cuts are necessary. Without negative back rake, there is danger of breaking off the tool point.

TABLE XI. DIMENSIONS FOR SHEAR TYPE TOOLS

A	B	C	T	W	L
1	1	7	5/16	7/16	15/16
1 1/4	1 1/4	8	3/8	5/8	1
1 1/2	1 1/2	8	3/8	3/4	1 1/4
1 1/2	2	8	1/2	3/4	1 1/2

Boring Tools. Boring tools are usually made in three types: the square shank, the round shank, and the solid carbide round shank. Square shank boring tools are shown in Fig. 13. Specifications for dimensions of these tools are given in Table XII. The round shank boring tools find wide application in industry. They are shown in Fig. 14 and their dimensions are given in Table XIII. Another type of round shank boring tool is shown in Fig. 15. These tools are made of solid carbide since it is not practical to make shanks in very small sizes. Dimensions for these tools are given in Table XIV. The solid carbide boring tools are made with angle A = 38° angle B = 30°; and angle A = 53°, angle B = 45°

TABLE XII. DIMENSIONS FOR STYLE 23, 27, AND 31 SQUARE SHANK BORING TOOLS

A	B	C	T	W	L	R
1/4	1/4	1 1/2	1/16	5/32	1/4	1/64
5/16	5/16	1 1/2	3/32	3/16	5/16	1/64
3/8	3/8	1 3/4	3/32	3/16	5/16	1/64
7/16	7/16	2 1/2	1/8	1/4	7/16	1/32
1/2	1/2	2 1/2	1/8	5/16	7/16	1/32
5/8	5/8	3	5/32	3/8	9/16	1/32
3/4	3/4	3 1/2	3/16	7/16	7/16	1/32

TABLE XIII. SPECIFICATIONS FOR ROUND SHANK BORING TOOLS

A	C	D	E	T	W	L
5/16	2	3/16	1/16	1/16	1/4	5/16
3/8	2	3/16	1/16	1/16	1/4	5/16
7/16	2 1/2	1/4	3/32	3/32	5/16	3/8
1/2	3	1/4	1/8	3/32	3/8	3/8

Several applications of boring operations are shown in Fig. 16. Modification of boring tool design is sometimes necessary to meet the needs for a particular job.

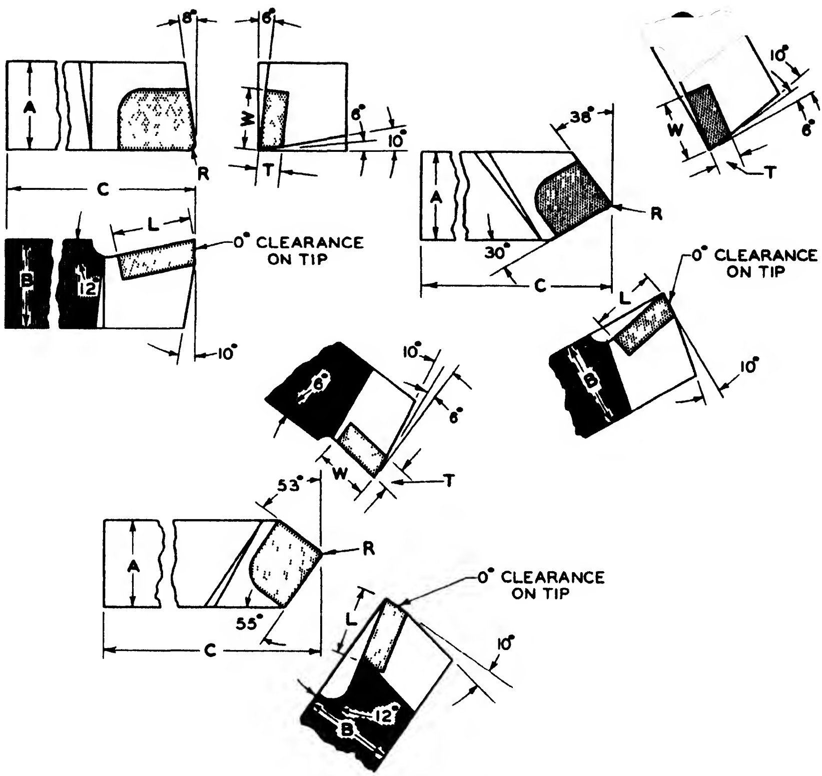


Fig. 13. Square Shank Boring Tools

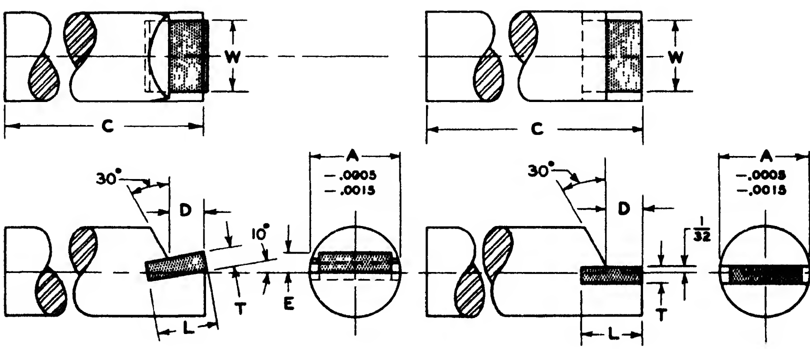


Fig. 14. Round Shank Boring Tools

Construction of Tools. In order that the tool tip will be accessible for grinding, the tool shank is milled, shaped, or otherwise recessed in such a way that the tip fits into the recess or seat and projects from it at the end, side, and top. The exaggerated drawing in Fig. 17 illustrates this point. This form of construction makes easier the grinding of the tool since it is necessary to grind only the carbide when sharpening the tool. Subsequent grinding, however, will make it necessary to remove some of the steel from the shank in order that the tip may be finished.

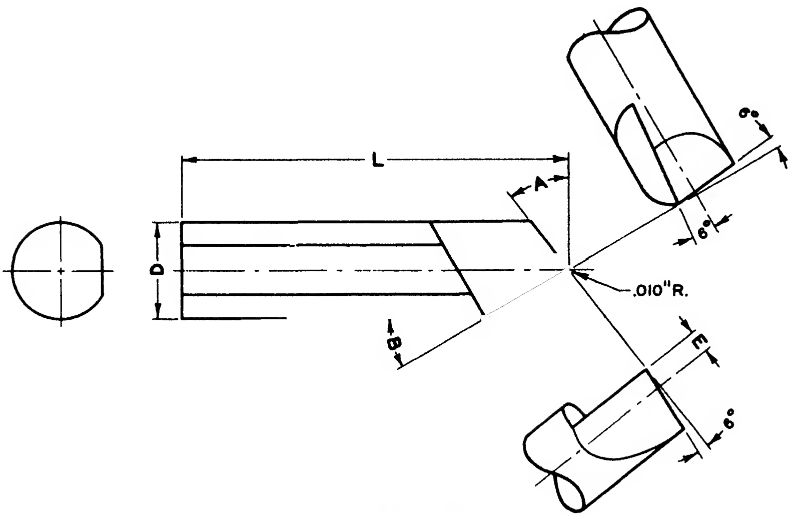


Fig. 15. Solid Carbide Boring Tool

In (A) of Fig. 18 is shown a typical shank design with a thin section of abutment material along one side of the tip, as shown at X. This design should be avoided since it has been found for some reason not yet fully explained or understood, that such construction produces more cracked tips in brazing and grinding than other types.

TABLE XIV. DIMENSIONS FOR SOLID CARBIDE BORING TOOLS

D	E	L
3/32	.015	3/8
1/8	.015	1/2
5/32	.015	5/8
3/16	.031	3/4
7/32	.031	7/8
1/4	.031	1

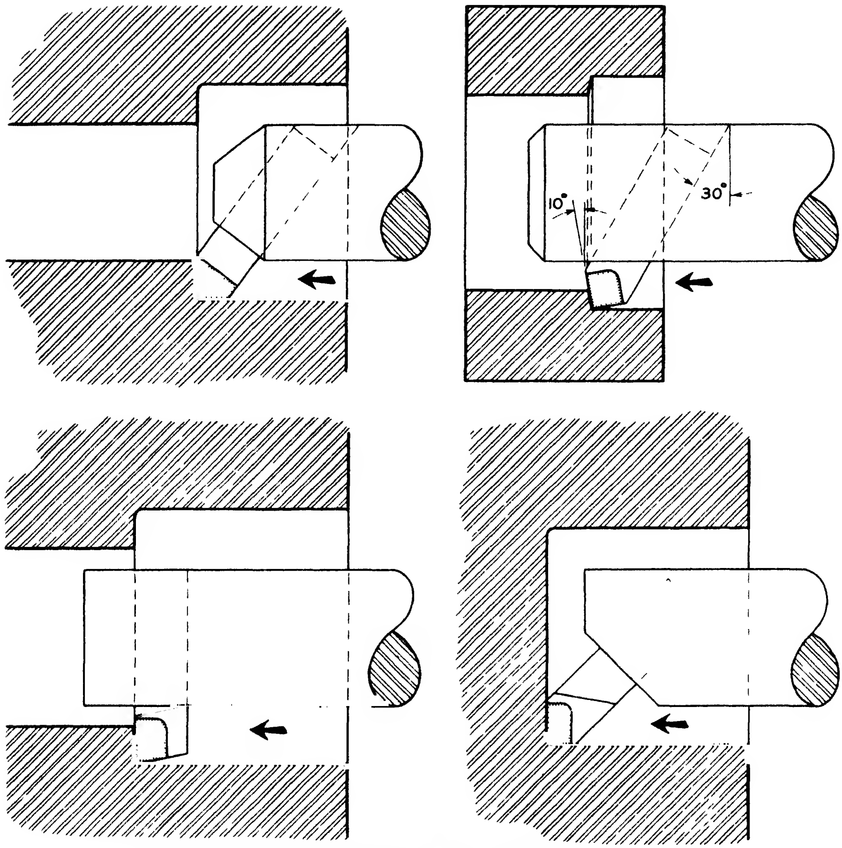


Fig. 16. Boring Bars and a Few of the Typical Operations Performed

An improvement over this design is shown in (B) of Fig. 18. This construction is calculated to reduce the possibility of cracked tips in brazing and grinding which result from the difference in thermal expansion of the steel shank and the carbide tip. The height of the recess in this design is reduced to one-third that of the thickness of blank used. The objectionable feature of this design is, however, that the strength of the shank has been reduced by the removal of material.

A tool shank with the tip recess milled through along the edge is shown in Fig. 19. This type of tool is easier to make than the shank with an end-milled tool seat such as that shown in (B) of Fig. 18. At the same time, it is just as effective, since the radial force seeking to push the tip out of the seat is not great enough to overcome the shearing strength of the brazed joint.

In (A) of Fig. 20 is illustrated a face grooving tool. It should be noted

that the supporting shank has converging curves on the sides. They enable the tool to clear the work and provide support for the tool tip.

A so-called "economy" tool is shown in (B) of Fig. 20. The distinctive feature of this design is the tip on each end of the shank. The only advantage of this design is the saving of a few cents per dozen on steel shanks which is more than nullified by the ease with which the tool can be damaged in setting up, handling, and use. It is very seldom that both ends of such a tool are used to the full extent.

When it is desirable to groove the work and at the same time chamfer the sides of the groove, a tool such as that shown in Fig. 20 at (C) can be used to advantage. However, this tool is actually a forming tool and will be discussed in greater detail in Chapter IX.

In (D) of Fig. 20 is illustrated a tool that can be used for turning, filleting, and facing of work in one operation. The objectionable feature of this tool is the narrowness of its neck which leads to the possibility of having the tip crack at the narrowest point. If a tool of this nature must be used, it should be sandwich-brazed with copper of 1/64" to 1/32" thickness.

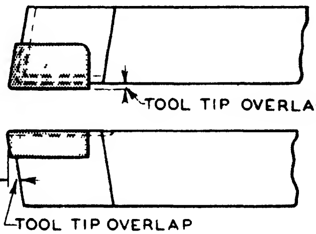


Fig. 17. Construction of the Tool So As To Facilitate Grinding

Clamp-Down Tools. Sintered carbide blanks or tips may be clamped as well as brazed to steel shanks, and used as cutting tools. Fig. 21 illustrates two tools made by Kennametal, Inc., with tips held in place mechanically. These tools can be used for turning, facing, and chamfering operations with depths of cut comparable to those of brazed-on carbide tip tools.

It is essential in this type of tool construction that the shank be properly recessed, hardened, and ground so that the tip will have solid support beneath it and against the abutment. If these conditions are not

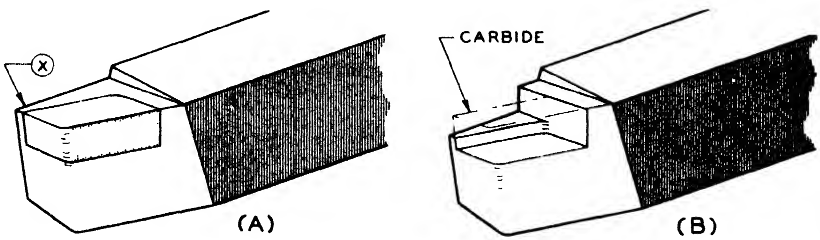


Fig. 18. Tool Shank with a Thin Abutment (A) and a Modified Design with Reduced Recess Height (B)

met, the tool, lacking the necessary solid support, may crack in the cutting operation. The function of the clamp or screw is to hold the tip in position. It does not absorb any part of the cutting forces. The clamp or

screw does not even have to be especially tight if the proper conditions of tip support are provided.

Other varieties of the mechanically held tip manufactured by the Kennametal organization are shown in Fig. 22. Dimensions for the shank and tip size are given on the drawing in (A) of Fig. 22 for the 12H190 clamped-on tool. The shank for this particular tool should be made of either a tool steel or an alloy steel in order that it can be hardened to from 36 to 38 on the Rockwell C scale.

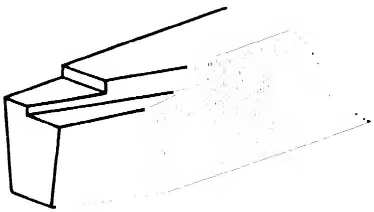


Fig. 19. A Tool Shank with the Recess Milled Through

In (B) of Fig. 22 is illustrated a tool holder which employs a round tool bit. This bit is ground concave on the top and bottom. When the edge is dulled, the bit is loosened in the holder and turned so as to bring the sharp part of the edge to the front for cutting.

In (C) of Fig. 22 is shown a tool holder which is fitted to hold a four-cornered bit. Since both ends of the bit are ground to sharp corners, there are eight cutting edges which become available merely by unclamping the tool, turning the bit 90°, and reclamping.

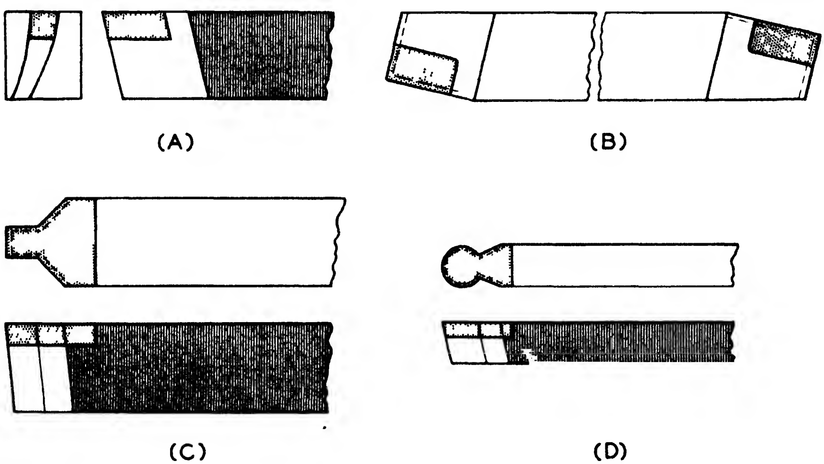


Fig. 20. A Face Grooving Tool (A), the Double-End "Economy Tool" (B), a Special Grooving and Chamfering Tool (C), a Forming Tool for Turning, Filleting, and Facing Operations (D)

In (D) is pictured a tool holder designed to use a triangular bit. This particular tool was designed for finishing cuts.

Tool-Holding Devices. For turret lathe work involving the use of single-point tools, a tool holder is often used such as the one shown

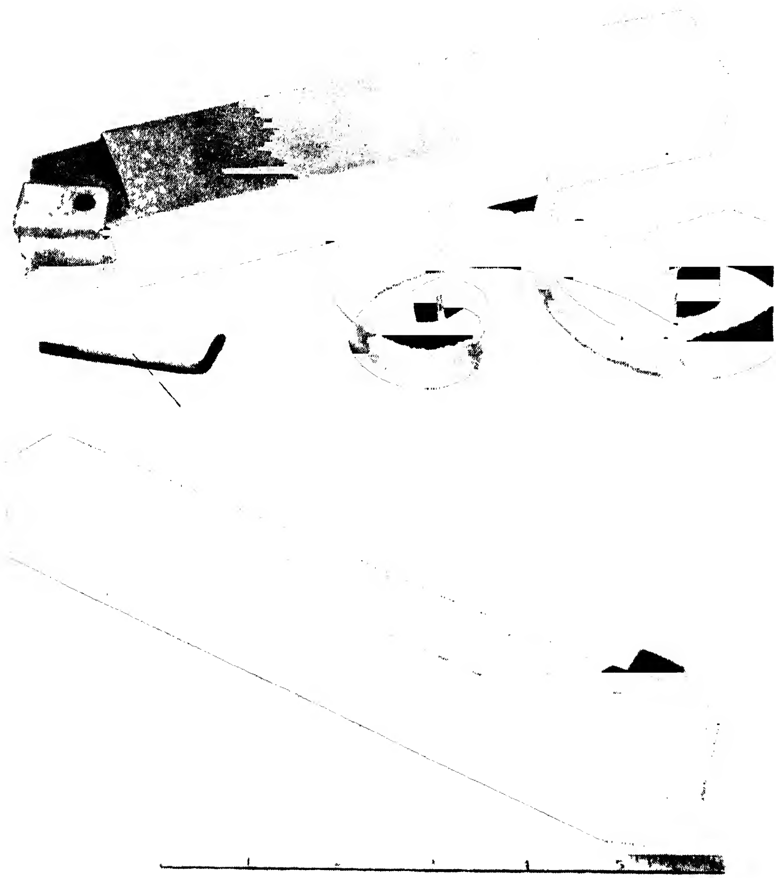


Fig. 21. Tool Holders into Which Carbide Tips Are Clamped or Screwed
Courtesy of Kennametal, Inc

in Fig. 23. The turret is used on the cross slide of the machine. Four tools can be set in the fixture. These can be used for finish facing, turning, chamfering, and grooving. Another type of setup for a turret lathe is shown in Fig. 24.

Fig. 25 illustrates a setup for turning the hub and the outside diameter of a gear blank with tools held in knee bars. The knee bars are held in a bracket which is attached to the main turret of the machine. As in all designs of and with carbide tools, the requirement here is rigidity. This is accomplished by using short, stubby bars as shown in the drawing.

Tool Blocks. For a single- or multiple-tool setup such as the one shown in Fig. 26, heavy-type block tool holders are used. These

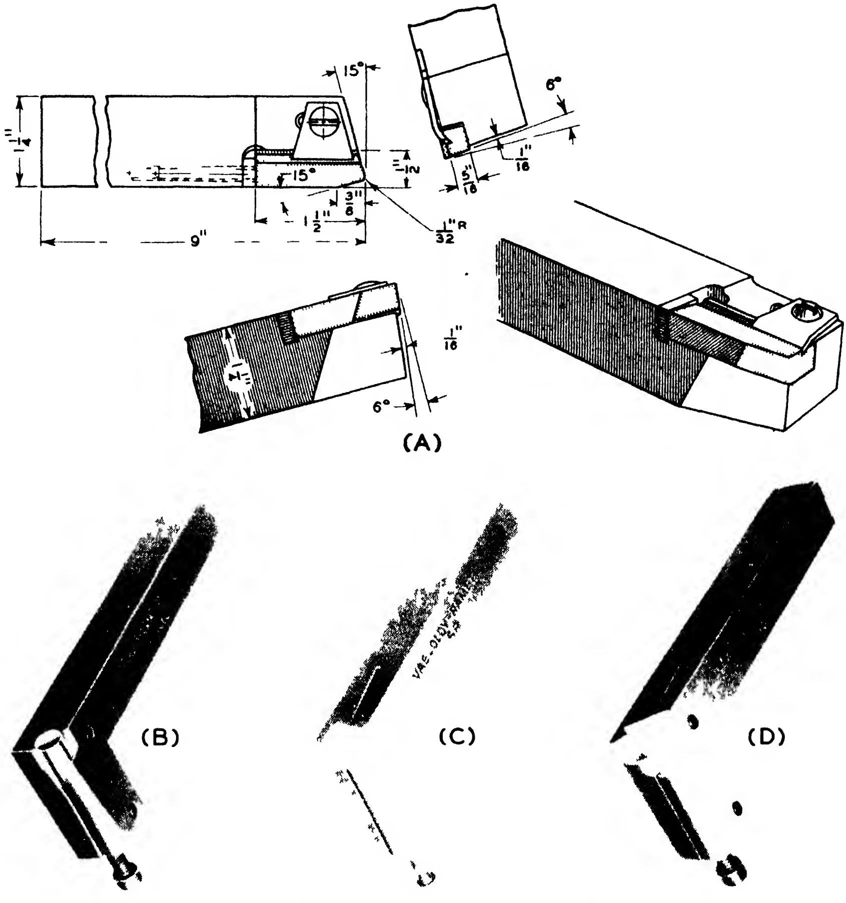


Fig. 22. Other Varieties of the Mechanically Held Tip

holders are rigid, having grooves in which the tools are set. The tools are adjusted for position by Allen type headless set screws. When the tools in this kind of a block are reground to the extent that the tip falls below the center line of the work, they can be brought back to the proper height with shims or by means of a pair of adjustable wedges under each tool.

Additional Examples of Tool Design. The information given concerning most types of single-point tools should be applied to additional cases of design problems in order to more fully understand the

points which have been described. For example, assume it is desired to machine the part shown in Fig. 27 in two cuts, roughing and finishing. The roughing cut can reduce the diameter by .170" at 250 f.p.m., while the finishing cut can remove .017" at 300 f.p.m. The feed for roughing may be .016" per revolution and the finishing cut feed .008" S.A.E. 3140 steel of 220 Brinell hardness is being cut. Determine the following unknowns:

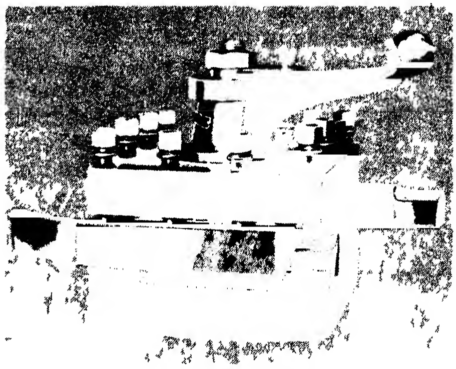


Fig 23 A Turret Tool Holder for the Cross Slide
Courtesy of the McCrosky Tool Corp

a) The force acting at the point of the roughing cut.

b) Whether a 10 hp motor will be enough to drive the machine if its efficiency is 75 per cent.

c) The thickness of the tool tip to be used for the roughing cut

d) The thickness of the tip for the finishing cut.

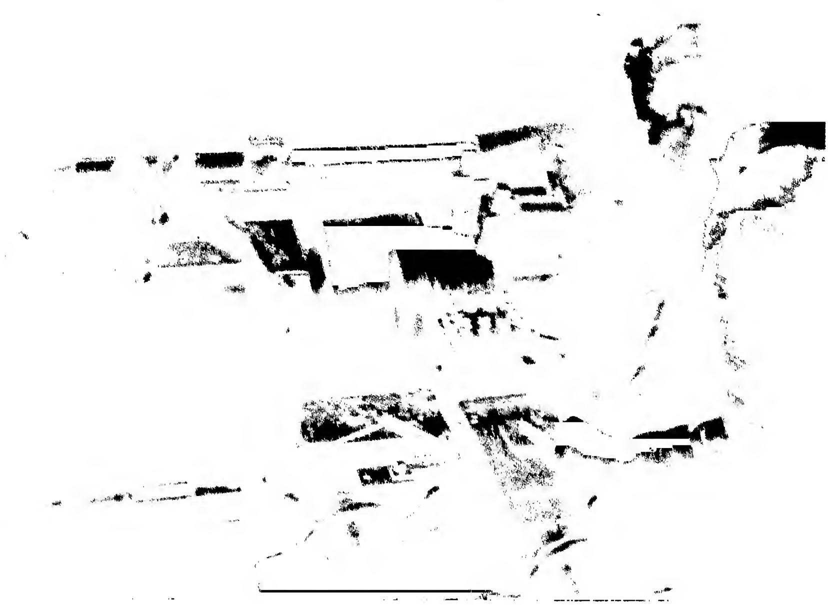


Fig. 24. Turret Lathe Setup in Which a Turret Tool Holder Is Used as Well as a Pilot Bar for the Overhanging Turning Tools

e) The size of the shank for the maximum deflection of .002" when the overhang of the tool is 2".

f) The style of the tool.

g) The design of the tool holder for the cut.

h) Whether the shank will stand the stress.

i) The size of the chip breaker.

Using the formula $P = K \times d \times f$ in determining the force at the point

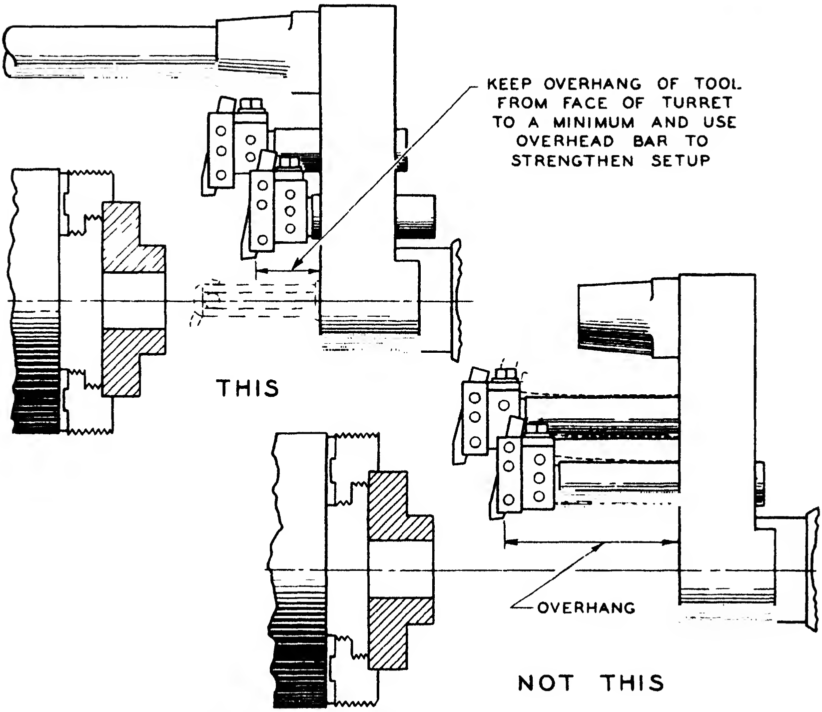


Fig. 25. A Typical Setup Using Knee Tools and a Pilot Bar

of the tool during the roughing cut, substitution of known values is made and K is taken at 350,000. The solution is, therefore:

a) $$P = 350{,}000 \times .085 \times .016 = 476 \text{ pounds}$$

b) Horsepower is figured from the formula,

$$hp = \frac{P \times V}{33{,}000 \times E}$$

Assuming the equipment is 75 per cent efficient and substituting the known values in the formula results in

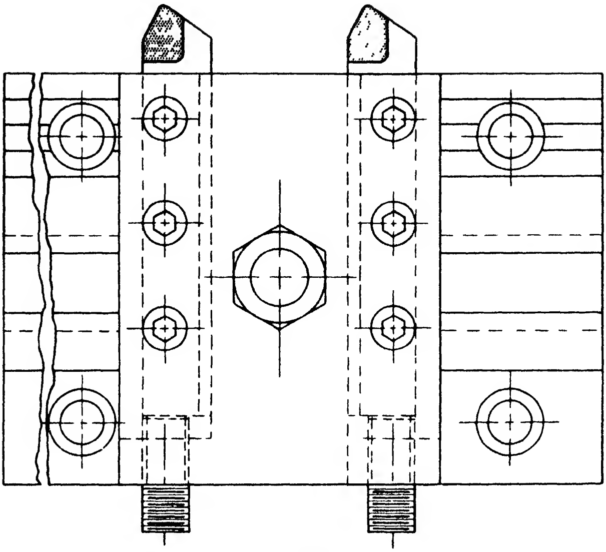


Fig. 26. A Block-Type Tool Holder for a Multiple-Point Setup

$$hp = \frac{475 \times 250}{33{,}000 \times .75} = 4.8 \text{ for one tool.}$$

For two tools used at the same time, the hp requirement would be 9.6, or approximately 10. A 10 hp motor would be adequate for the job.

c) The thickness of the tip can be selected from Table IV. For a 3/32" depth of cut and a .016" feed, the tip should be at least 5/32" thick. It would be well to use the next larger size so as to allow for a few regrinds before reducing the effective thickness of the tip. Hence, a 3/16" or even a 1/4" tip should be used. However, the most conservative choice from the table is the 3/16" thickness.

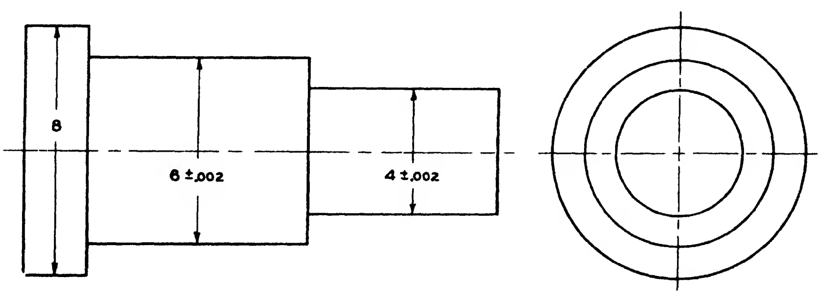


Fig. 27. Example of Machine Part Illustrating the Machining Problem

d) Since the tools for the roughing and finishing operations will be similar, the same type of tool holder can be used for both tools. For this reason, it will be more economical to specify the use of the same size blank and shank for the finishing tool and for the roughing tool.

e) The size of the shank should be selected tentatively from Table

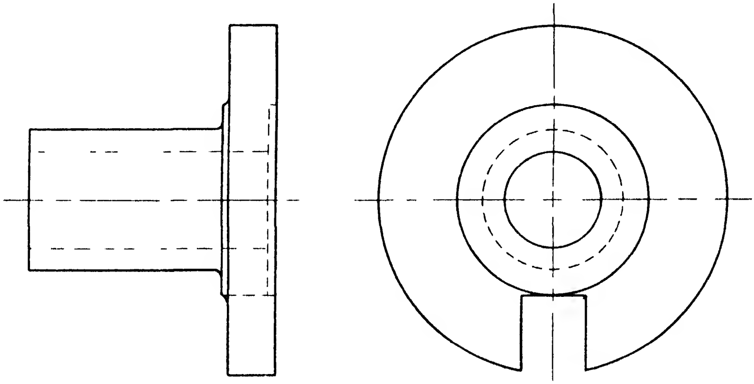


Fig. 28. A Second Example Illustrating a Problem in Machining

VI. On this basis, a 3/4'' × 1'' shank could be chosen, subject to design changes. Allowing a 2'' overhang, the tool must be checked for deflection. The formula used was given in the section of this chapter describing the design for shank deflection. The known values are P = 476, L = 2, E = 30,000,000, b = 3/4, and h = 1. Substituting these values in the formula gives

$$y = \frac{4 \times 476 \times 2 \times 2 \times 2}{30,000,000 \times 0.75 \times 1 \times 1 \times 1} = .0007 \text{ nearly}$$

This is less than the .002 specified for the tool and indicates that the shank selected is rigid enough for the cut. A smaller shank might be used, but it should be carefully checked to make certain its deflection is within the prescribed limits.

f) From an examination of the work, it is evident that tool styles 1 and 3 in Fig. 6 probably are most suitable for the job.

g) The tool holder might well be of the type shown in Fig. 26, with adjusting screws against the shank as shown in the figure.

h) The stress can be checked by substituting the known values for the formula given in the section of this chapter on strength and deflection. This is as follows:

$$S = \frac{6 \times P \times L}{b \times h} = \frac{6 \times 476 \times 2}{0.75 \times 1 \times 1} = 7,600 \text{ p.s.i.}$$

This is fairly low and may represent a factor of safety of 10 or more, depending on the strength of the shank material.

(i) The function of chip breakers and chip curlers was thoroughly discussed in Chapter V. Table VII in that chapter gives recommended depths for chip breakers under varying conditions of use. For a cut .085'' deep with a .016'' feed, the width of the chip breaker should be at least 1/8'' and its depth from .010'' to .015''.

Assume it is desired to face two sides of a flange at the same time. The cut on each side is to be 1/16'' deep, and the feed selected is .012'' per revolution. The material to be cut is a low-alloy steel forging with a hardness of 200 Brinell. The cutting speed for an interrupted cut on the piece, shown in Fig. 28, is 225 f.p.m. The tool overhang is 2 1/2''. Determine the following unknowns:

a) The force on each tool. (Assume K = 275,000)

b) The horsepower necessary to drive the machine if its efficiency is 70 per cent.

c) The size of the tip for the job.

d) The size of a square shank if the allowable deflection is not to be greater than .001''.

e) Whether the stress in the shank is within permissible limits.

Using the formula given earlier in the chapter for determining the forces working upon a tool, values are substituted for letters resulting in the following solution:

a) $$P = 275{,}000 \times 0.062 \times 0.012 = 205 \text{ pounds}$$

b) The horsepower necessary is

$$hp = \frac{205 \times 225 \times 2}{33{,}000 \times .70} = 4$$

This is for two tools.

c) The size of the carbide blank for the tip is selected from Table III in this chapter. For a cut of this size, a tip 1/8'' in thickness is required. However, for an interrupted cut, a 3/16'' × 3/8'' by 3/4'' blank would be necessary.

d) The size of the shank suitable for this job can be computed with the formula given earlier in the chapter for determining shank deflection. Since a square shank is to be used, h can be substituted for b. The formula can then be solved for h thus:

$$y = \frac{P \times L^3 \times 4}{E \times b \times h^3} \quad \text{for rectangular shanks}$$

$$y = \frac{P \times L^3 \times 4}{E \times h^4} \quad \text{for square shanks}$$

$$h = \sqrt[4]{\frac{P \times L^3 \times 4}{E \times y}}\,^{*} \quad \text{for square shanks}$$

Substituting 30,000,000 for E and the proper values for the other known letters in the formula, results in

$$h = \sqrt[4]{\frac{205 \times 2.5 \times 2.5 \times 2.5 \times 4}{30{,}000{,}000 \times .001}} = \sqrt[4]{0.4204} = 0.8$$

This answer indicates that a 7/8'' square shank would satisfy all the conditions. However, a shank 1'' square would probably be a better choice.

e) The stress would be figured using the formula given in an earlier section of the chapter, substituting values for the letters in the formula. Thus,

$$S = \frac{6 \times 205 \times 2.5}{1 \times 1 \times 1 \times b \times h^2}$$

or 3,100 p.s.i., approximately. This is fairly low.

In addition, it is well to remember this simple rule: In all cases of tool design with cemented carbide tips, the shank should be not less than three times the thickness of the tip, and preferably more to make rigidity a certainty.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Define a "standard" blank for a tool tip.
2. Define a "standard" shank.
3. Define a "stock" tool.
4. Define "offset turning tool" and state its purpose.
5. Can a positive back rake angle tool be used for interrupted cuts? Explain your answer.
6. When is a negative back rake tool recommended?
7. Can the rigidity of a steel shank be increased by hardening it?
8. Can the strength of a shank be increased by hardening?
9. What is the function of a chip breaker?
10. Is a chip breaker necessary for turning brass, bronze, and cast iron? Why?
11. Select the size of the blank for a tip to take a 1/4'' deep cut at a 1/16'' feed per revolution.

*To find the fourth root, use logarithms or extract the square root twice.

12. A 1/4" deep cut is to be taken on soft steel. The feed is 1/32" and the cutting speed 300 f.p.m. Determine the pressure on the tool and the horsepower necessary to make the cut.

13. If the steel shank in the last problem is to be twice as high as it is thick, that is h = 2b, what should be its size for a possible overhang of 3" and a permissible deflection stress of 8,000 p.s.i.? Suggestion:

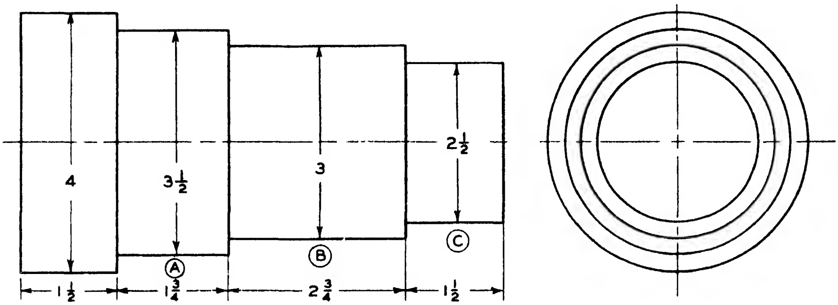


Fig. 29. A Machined Part Requiring Three Tools in Turning
Courtesy of Warner & Swasey Co.

Use a variation of the formula given for solving the resistance moment, and solve for b.

14. If the tool shank in the last problem were limited to 2" overhang and the permissible stress were increased to 10,000 p.s.i., what would be the size of a square shank; that is, when b = h?

15. By heat treating the square shank, its strength was increased to 175,000 p.s.i. Using a factor of safety of ten, what could be the design stress, and what should be the size of the shank in question 14?

16. Design a boring bar with a boring tool to cut a 2" rough hole. (Suggestion: The boring bar may be 1 3/4" in diameter and the tool shank 3/8" to 7/16" round.)

17. If the bar in question 16 is to project from the support 6" and the cut is to be 1/16" deep with a .012" feed per revolution, what will be its deflection if the cut is to be made in a low-alloy steel forging? (Suggestion: Find the pressure on the tool, using 350,000 for K. Then use the formula given for computing deflection. Since the bar is round, substitute for I the moment of inertia,

$$\frac{3.1416 \times D^4}{64}$$

where D is the diameter of the boring bar.)

18. Design the bar in question 17, using carbon tool steel. Draw two views, showing the clamping and adjusting screws in place. Specify hardening, drawing or tempering at 700°F., and grinding after hardening.

19. Design a block-type tool holder for holding in their proper relationships, three tools for turning the piece shown in Fig. 29.

20. Design a wedge-type shim for elevating the tools in question 19 after they have been repeatedly ground on the top and fall below the center of the work.

REFERENCES

Faires, V. M. Design of Machine Elements. New York: MacMillan Co., 1938.

Kennicott, W. L. "Mounting Solid Cemented Carbide Cutting Blades Mechanically," Mechanical Engineering, Vol. 67, 1945.

Warner & Swasey Co. "Turret Lathe Tools."

Vascoloy-Ramet, "Tool Engineering Data."

CHAPTER VIII

Coolants and Carbide Tools

Importance of Care and Handling. The sintered carbides, as has been pointed out, are extremely hard, having great compressive strength, though low tensile strength. They lack ductility, tend to crumble at the edges, and sometimes check during grinding or machining operations. Because of these characteristics, special methods of grinding and special techniques in machining must be employed in order to obtain the best results. To achieve this end, a special shop routine should be followed. A study of the suggestions in this chapter will help in the establishing of such a routine and will aid in avoiding many costly mistakes.

The performance of the tool, so far as its life and the finish obtained are concerned, depends in a large measure on the finish of the tool. For this reason, the tools must be sharpened by special methods on carefully selected wheels which are frequently of a very fine grit size. This is necessary in order to obtain a fine cutting edge. The methods of grinding and finishing the single-point tool were discussed in Chapter V. Perhaps it would be well at this time for the reader to review the recommendations made there.

Selection of Proper Carbide Grade. Sintered carbides are made in many grades. Some of them are straight tungsten carbides. In general, these are suitable for cutting cast iron, malleable cast iron, semisteel, and nonferrous materials. Other grades are a combination of tungsten carbide with tantalum-columbium carbide or titanium carbide, and sometimes two or more of these combinations are used together. The binding material is usually cobalt, occasionally nickel, and, to some extent, iron. These grades are used in machining steel and steel castings. Different grades, varying in hardness, have been developed by the various companies making sintered carbide blanks for tool tips. The literature of these companies often contains much worthwhile information. For soft, ductile materials like screw stock, a softer grade of sintered carbides is recommended. For heavy, interrupted cuts or heavy hogging cuts in soft materials, softer but stronger grades of carbide are best. For tough materials such as alloy steels, harder grades are used. Finally, for hard metals, for finishing cuts at high speeds and low feeds, abrasion and cratering resistant grades are recommended. These are often of a hardness of 93 on the Rockwell A scale.

Information was given in Chapter II concerning the conversion to the use of carbides. Detailed data regarding the selection of the proper carbide grades is given in Chapter XV. However, these points will be touched upon here for purposes of review and emphasis.

Selection of the proper grade of carbide for a job is of prime importance because the all-around performance of the tool as well as its life are dependent upon the use to which it will be put. While it is true that all carbides will cut, regardless of the material to be machined, their performance is another matter. For example, a grade suitable for cast iron will also cut steel, but because its resistance to cratering is low, it will not perform satisfactorily. Conversely, the grades suitable for machining steel will also cut cast iron but the tool life and the finish obtained will not be satisfactory.

Selection of Size and Design. Since all carbide tools operate at higher speeds than do the high-speed-steel tools, they remove more material per minute than do other types of tools. Cutting feeds and depth of cut are frequently greater than those taken with high-speed steel. Since, in most cases, part of the shank is cut away to accommodate the carbide tip, the shank is somewhat weaker than that for the corresponding size of high-speed-steel tool. These factors require that the carbide tool shank be larger and deeper. Only then can an entirely satisfactory performance be expected.

Consideration also must be given to the thickness of the tip appropriate for the job. This was discussed in Chapter VII. Recommendations for depths of cut and rates of feed were given in Table V. For example, it would be false economy to take a 1/2" deep cut in steel with a feed of .032" per revolution, using a tool having a tip 1/8" thick. A tip 3/8" to 1/2" in thickness should be used for such a cut. It should be remembered that the carbide tool tip will decrease in thickness with each regrinding. When the tip becomes too thin to take the cut it was originally designed for, it should be put aside for use on smaller cuts.

Dimensions for standard shank sizes also were given in Chapter VII. This information was contained in Table VI. Once the design is selected or modified, it should be checked against the cut for stress and deflection. Otherwise, it may not give satisfactory performance. The shank, in all cases, should be used with the greatest height against the cut to enable it to resist deflection under the cut. This principle is shown graphically in Fig. 1 at (A) and (B).

Selection and Inspection of Machine. Modern machines have been constructed rugged enough to take a good cut with carbide tools. They have sufficient power and speed, substantial bearings, and are free from vibration. However, it is necessary to check an older machine for speed and power for each particular job since these are the factors necessary for the successful application of sintered carbide tools. There should be enough power so that the machine will not stall under the cut. Obviously, the higher the speed for a given cut, the greater will be the demand for power. Speed is a prerequisite to the use of carbides,

particularly when working on soft, stringy metals such as low-carbon and low-alloy steels where the built-up edge interferes with proper cutting and results in poor finish and short tool life. Evidence of the best conditions of speed is shown when the finish obtained is of a satin tex-

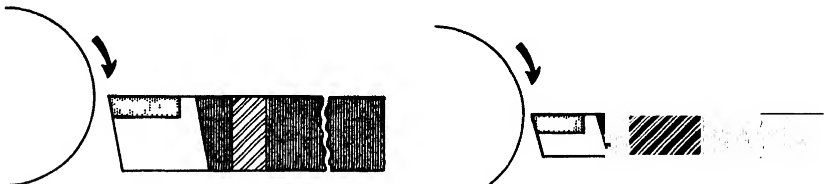


Fig. 1. Illustrating the Correct and Incorrect Method of Using the Rectangular Shank

ture, without glaze. It is at this speed that tool life can be expected to be maximum. On the other hand, excessive speeds will shorten the life of the tool between grinds.

It is not necessary that a machine be new for sintered carbide application, but it should be in good condition and have ample speed and power. Before putting sintered carbide tools to work on an old machine, a check should be made for excessive play in the spindle bearings and in the slides. As was pointed out in Chapter II, the driving mechanism should be of the type that does not slip under the cut. When older machines are converted to the use of carbide tools, they should be modernized by the installation of more powerful motors and multiple V belt drives. This will provide a low-cost conversion in terms of benefits obtained.

Geared-head machines of the older vintages have a tendency to gear chatter—the rhythmic vibration produced by the revolving gears themselves. Although these vibrations are small, they are transmitted to spindles and to the work and invariably contribute to an early failure of the tool. It is necessary in such cases to have spindle bearings well fitted to minimize the effects of gear chatter on the work.

Small, fractional-horsepower-driven motors can be used for carbide applications, but the depth of cut and feed obviously should be conservative so as not to overload and stall the machine.

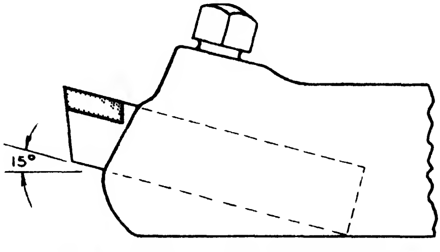


Fig. 2. The Holder for a Tool of High-Speed Steel

Tool Holders. The high-speed-steel tool holders of the type shown in Fig. 2 do not lend themselves to successful use with carbide tools. This holder tilts the tool at 15° as shown in the drawing, and requires a

front clearance of 21° to 22°. This would result in a very small lip angle on the tool with a thin cutting edge as shown in Fig. 3. This type of cutting edge is not suitable for the carbides. Tool holders of this type can be used with carbide tipped tools, however, provided they hold

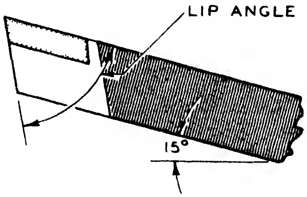


Fig. 3. A Thin Cutting Edge is Undesirable in the Carbides

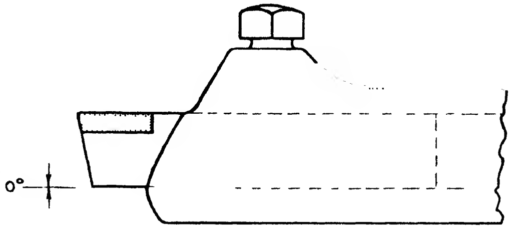
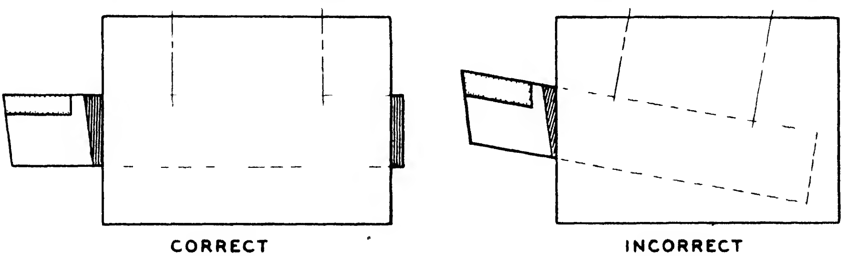


Fig. 4. Illustrating Use Requirements of Tool Holder with Carbide Tools

the tool in a horizontal position as shown in Fig. 4. It should be noted here that the back rake on the tool is zero.

In (A) and (B) of Fig. 5 is shown the right and the wrong way to mount the tool in the block-type tool holder when machining steel and cast iron. When turning aluminum and magnesium, it is desirable to give a slight amount of back rake to the tool. This may be accomplished by holding the tool at an angle in the tool holder. An exaggerated drawing of this situation is shown in (B) of Fig. 5.

Another important item which cannot be overemphasized is the permissible amount of unsupported overhang of the carbide tool. Too much overhang, as at (A) in Fig. 6, is detrimental to the tool because deflection increases rapidly in proportion to the overhang. If the tool projects just a little more than usual, it would be well to add additional support to the shank as shown in (B) of Fig. 6. Where it is not practi-



CORRECT
(A)

INCORRECT
(B)

Fig. 5. The Right and Wrong Way To Hold Tools in the Block-Type Tool Holder

cable to have additional support, the tool should be more rigid when the overhang is greater. A good rule is to keep the overhang to not more than one-half the height of the tool shank as shown in (B) of Fig. 6. Dimension X, if possible, should never be greater than dimension Y for average applications.

The tool holder itself should be of substantial construction and should be well clamped to the slide for rigidity. Except for occasional use with very small carbide tools and in the cutting of light metals, special tool holders for use with carbides are generally of little value.

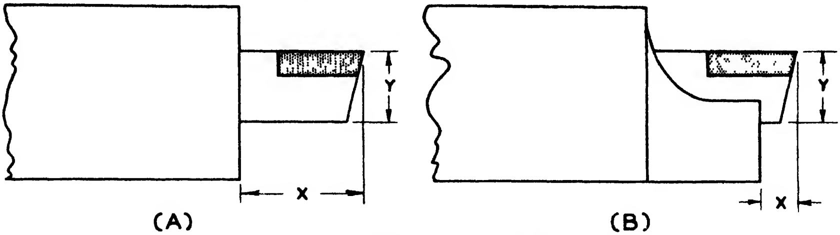


Fig. 6. X Should Be No Greater than Y for Average Conditions

It is quite important in turning operations that the tools be placed at the center of the work. This is clearly shown in Fig. 7. When the tool has been resharpened and its cutting point falls below the center line of the work, the cutting point must be elevated by shimming the shank. Elevation of the tool should never be done by "rocking" it in the tool post of the type shown in Figs. 2 and 4. Although this method is employed where high-speed-steel tool bits are used, rocking a carbide tool will change the front relief angle as well as its back rake and may contribute to early tool failure.

Cutting Angles. Sintered carbide tools, as well as all other cutting tools, should have just enough relief to provide cutting action which is free of rubbing. The side rake and back rake angles should also be maintained by correct grinding. Relief and cutting angles were discussed in Chapter III and a table of relief or clearance angles was given together with cutting angles for different kinds of metals. These angles can be maintained by proper setting of the tool.

Relief angles for the work should be held at the minimum so that there is ample support for the cutting edge. If excessive angles are used, the tool may not stand up as well as it should. When regrinding the tools, use the straight cup wheel for finishing. The primary and secondary relief angles should be ground as shown in Fig. 8.

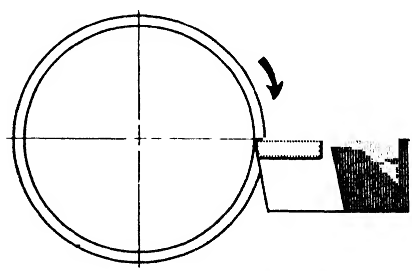


Fig. 7. The Tool Should Be Placed at the Center of the Work

When it becomes necessary to grind the relief angles on the outside diameter of the wheel, the relief on the tool becomes concave. An exaggerated drawing of this is given in Fig. 9. The concavity may be partially eliminated by holding the tool at an angle to the face of the wheel. In such grinding, however, it is desirable to use a grinding wheel

which is at least 16" in diameter. If this is impossible, the tool should be finished on a cup wheel after having been roughed out on the straight wheel. This will assure the right amount of tool support as was shown in Fig. 8.

Regrinding of tools should be done wet when the equipment is available. It is essential under wet grinding conditions that the tool receive a continuous and copious supply of coolant during the grinding. Otherwise, the intermittent cooling and heating may result in the cracking of the tool.

Dry grinding of carbide tools may be accomplished, but extra care must be exercised so as not to overheat the tool. The silicon carbide wheel should be dressed often when grinding dry so as to provide free cutting action. If the wheel is dressed frequently, it is not likely to overheat the tip. When the tool becomes hot during grinding, it should not be dipped in water or any other liquid as a means of cooling. Such action invariably results in a cracked tool. The tool should be allowed to cool slowly in the air.

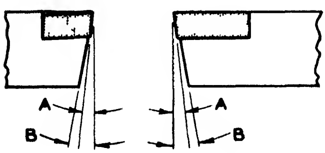


Fig. 8. Correct Primary and Secondary Reliefs

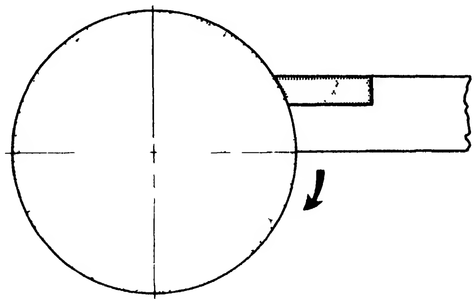


Fig. 9. Grinding the Tool on the Straight Wheel May Make the Point Concave

How to Handle Tools. When carbide tools are being used, the machine is started in the same way as for cutting with high-speed steel. It is in stopping the machine that the technique differs. To stop the machine while the cut is being taken, it is necessary first to disengage the feed. If this is not done, a chipped tool point will result.

Should the cut stall the machine, the tool should not be pulled from the cut nor the work reversed to free it. Instead, it is necessary to loosen the screws holding the tool in the tool holder. The tool is then removed through slow reverse turning of the work. At the same time the cross slide is carefully moved away. Unless these precautions are observed, a broken tool point will result.

When the machine in operation develops chatter or vibration of the spindle or the work, it should be stopped immediately and the cause removed. This may necessitate the refitting of spindle bearings and possibly the cross slide on older machines or on some much-used newer ones.

When chatter is the result of slenderness of the work, the condition may be remedied by giving the work better support by using a steady or follow rest. Older machines may chatter when they are operating in back gears. To avoid this, they should be run on open belt whenever possible. Chatter of the part that is being machined may be reduced or eliminated through reduction of tool-surface contact with the work by grinding a smaller radius on the tool nose. Another way is to set the tool so that the cutting force is less likely to spring the work piece. In some cases, chatter may be eliminated by changing the speed and the feed. In other cases it may be ended by changing the relief angles or the back and side rake angles.

Chatter may be the result of excessive overhang of the tool and too much deflection of the shank under the pressure of the cut. For this reason, it is well to have the shank rigid and with as little overhang as practicable. Should it become necessary to have a greater overhang than the height of the tool shank, additional support should be provided for the shank.

Tool Life between Grinds. How long a tool should be kept working is a question frequently asked by machine operators and mechanics. The reason for this is the durability and long life of the tools which fosters the tendency to keep them in the machine as long as they will cut. This is false economy, for the rate of tool wear is greater once the tool starts wearing beyond certain limits. Tool pressures increase, more heat is generated at the tool points, and there is more likelihood of excessive cratering and wear of the points. The dull tool will invariably destroy the accuracy of the work and may even cause the springing of it.

In progressive, economy-minded shops, the tool is removed when it shows signs of wear, indicated by the increase in size of the work. This increase in size for roughing operations may be as much as .004", which corresponds to flank wear at the point of the tool of about .018" for 6° front relief angle, measured in the downward direction. The tool is then removed from the machine and retouched or reground by removing the minimum amount of material from the cutting edges. For finishing operations, the removal of the tool may be necessary when the accuracy of the work has been impaired as little as .0005". In many plants, the tool is removed for reconditioning after it has taken a specified number of cuts whether it appears to need it or not. This is often sound economy because the tool will be sharpened by removing only a few thousandths of an inch of carbide from the cutting edges. Such practice will give more pieces machined per tool instead of merely more pieces per grind.

Economical Speeds and Feeds. Speeds, feeds, and depths of cut for any set of machining conditions determine the efficiency and the rate of machine production. Tool life is influenced more by cutting speed and feed than by the depth of the cut. For this reason, a deep cut with high speed and low feed rate should be used. In taking cuts at

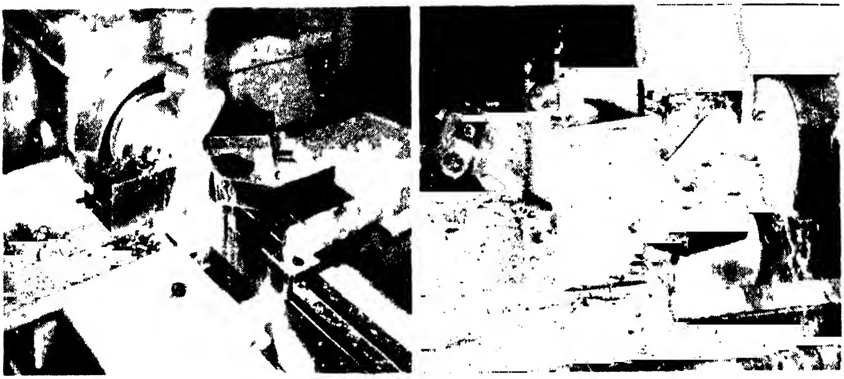


Fig. 10. Accepted Methods of Applying Cutting Fluids
Left, Courtesy of Warner & Swasey Co
Right, Courtesy of Sundstrand Machine Tool Co

lesser feeds, the pressure, which is independent of the cutting speed, is small. Therefore, deflection of the tool and the work is minimized, and chatter and breakage of the tool tip will be prevented.

By using high speeds and light feeds, excellent machine finish and high production rates are achieved. Furthermore, the cutting of metal at high speeds and light feeds imposes less strain on the machine and minimizes the wear and tear on moving parts and bearing surfaces. When carbide tools are operated at low speeds and coarse feeds, poor finish and shortened tool life may be expected. Recommended speeds and feeds were discussed in detail in Chapter VI. Here, it is necessary only to say that it is well to start at a conservative speed and feed, and vary the operating conditions until the best results are obtained. A speed which is too high will wear tool edges excessively.

When medium speeds and heavy feeds are used in machining strong and stringy materials such as alloy steels, there is much danger of tool failure through cratering. The amount of cratering may be lessened by an increase of speed and reduction of feed. With speeds which are too low, on the other hand, there will be an excessive amount of built-up edge on the tool point. This results in poor finish of the work and, frequently, in the chipping of the tool edge. This condition can be remedied by speeding up the work.

Cutting Fluids. Machining of metallic parts can be done either wet or dry. When the equipment is such that it is difficult to supply a continuous and copious amount of cooling liquid to the point of the tool, the cutting should be done dry. Insufficient or intermittent cooling of the carbide tool will invariably ruin it.

When cutting wet, the speed may be increased for the same tool life as much as 25 per cent or more. By the same token, the tool life will be increased materially when the cutting speed remains the same. In most cases of high production work such as screw machine or turret lathe

operations, the cutting is done wet, the proper cutting fluid being used for the operation. Typical applications of cutting fluid are shown in Fig. 10.

Cutting fluids are usually mineral oil blended with animal oil, or soluble mineral oil dissolved in water in a ratio of from 1 to 20 to 1 to 40 by weight (20 parts of water to 1 part of oil).

Sulphurized oil is used extensively in screw machine work, partic-

TABLE I. SUGGESTED CUTTING FLUIDS FOR TURNING, MILLING, AND REAMING

Material to Be Cut	Turning	Milling	Reaming
Aluminum	soluble oil mineral oil with 10% fat	soluble oil mineral oil	mineral oil with 10% fat oil
Alloy steels	sulphur base oil mineral oil with sulphur base (oil 75 to 25)	mineral oil with lard oil (90 to 10)	same as for turning
Tool steel Low-carbon steel	mineral oil with lard oil (75 to 25)	soluble oil	same as for turning
Brass	mineral oil with 10% fat	soluble oil	soluble oil mineral oil
Bronze	soluble oil	soluble oil	soluble oil
Copper	soluble oil	soluble oil	soluble oil
Monel metal	soluble oil	soluble oil	mineral oil with fats (25 to 75)
Malleable cast iron	soluble oil	soluble oil	soluble oil
Magnesium	mineral oil high flash point	mineral seal oil high flash point	mineral oil high flash point
Grey cast iron	dry or soluble oil	dry or soluble oil	dry or soluble oil

ularly in automatic screw machines, since the oil has a lubricating effect on the slides and the cams. Some authorities object to the use of sulphurized oil in connection with carbide tools, asserting that sulphur attacks the binder in the carbide and disintegrates it. There is no valid

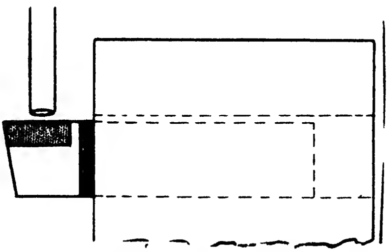


Fig. 11. Coolant Applied from Above

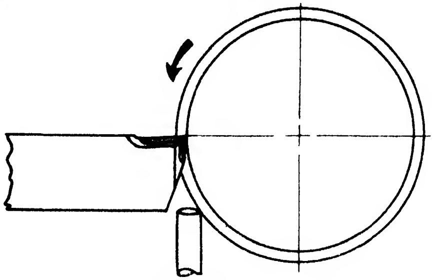


Fig 12 Coolant Applied from Below

proof that such is the case. However, there is some ground for suspicion that sulphur, when suspended in the oil, may be detrimental to the carbides and also to high-speed-steel tools. Sulphurized oil, with the oil in an active state (that is, in combination with the oil), is the best cooling medium yet found for carbide applications in automatic screw machine work.

In general, cutting fluids may be classified into four groups:

1. Conventional soluble oils—those that are emulsions of straight petroleum oils.

2. Super-soluble oils which are emulsions of fats, fatty petroleum mixtures, or other high-cutting quality oils.

3. Low-sulphur cutting oils containing sulphur in a chemically active state, with the percentage of sulphur less than 2.

4. High-sulphur cutting oils containing sulphur in a chemically active state, with the percentage of sulphur greater than 2.

Selection of a Cutting Fluid. Emulsified oil mixtures are used for common, ordinary work. When soft, tough, draggy materials are encountered, the straight oils are helpful in eliminating the built-up edge and the tearing of the material.

The low-sulphur oils work best on free-cutting stocks such as free-cutting steels, high-carbon steels, tool steels, steels of higher Brinell hardness, and nonferrous metals.

The high-sulphur oils are used on tough metals such as stainless steels, alloy steels, Monel metal, nickel, copper, tough bronze, steels with very low Brinell readings, and wrought iron.

The oil companies which manufacture and market these cutting oils have accumulated a great fund of useful information pertaining to their application. They are in a position, therefore, to render much valuable assistance regarding the technical problems involving the use of cutting oils. Table I will serve as a starting point in the selection of these coolants.

Methods of Application. For successful cutting, the fluid should be shot with sufficient force to provide complete immersion of the tool during the entire time it is cutting. This should be done in such a way that the chips do not interfere with the flow. One method is to direct the coolant through a pipe from above as is shown in Fig. 11. This method has one disadvantage in that the liquid is pushed away by the fast flowing chips, thus cooling the tool point less than it should.

Another method of using a coolant is to direct the flow under the tool as shown in Fig. 12. The advantage to this method is that the tool shank under the tip is also kept cool. Some tool engineers prefer to throw a double stream from both sides of the tool which converges on the tool point. This method is illustrated in Fig. 13.

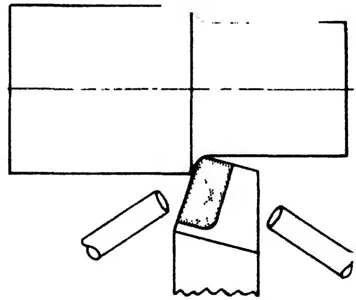


Fig. 13. Coolant Directed against Work and Tip from Both Sides

Whatever method of cooling is used, it is essential that the stream of coolant be directed on the cutting point. It is equally important that the stream be adequate, 2 to 3 gallons per minute being considered sufficient.

Handling Carbide Tools. Sintered carbide tools should be handled with extreme care. They should not be permitted to drop on the floor or come in contact with metals or stones, save when cutting or being ground. Single-point cutting tools should be kept in wooden boxes, each tool in a separate compartment. When this cannot be done, it is possible to protect the cutting edges from chipping by taping them with ordinary friction tape. Small boring tools may be protected from damage by taping the point, or providing them with slipover rubber caps. Very small boring tools may be attached to the tape and wrapped in it.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What are the characteristics of sintered carbides?
2. What happens when an improper grade of carbide is selected for a cutting tool?
3. What size blank should be used for a tool tip to take a cut 3/8" in depth and at .025" feed?
4. What would be an economical speed for cutting machine steel of 185 Brinell hardness?
5. Why is it so important to check the machine for power before starting a job with sintered carbide tools?

6. What is the effect of a loose bearing in the machine on the life of a tungsten carbide cutting tool?

7. What are the principal causes of chatter of the work?

8. Why is it uneconomical to permit the tool to wear down too much between grinds?

9. What is the function of a cutting fluid?

10. What are the three methods of applying cutting fluid to the tool?

11. Can tool life be increased by the use of cutting fluids?

12. Why do sintered carbide tools require more careful handling than tools made of high-speed steel or carbon-tool steel?

13. What should be the relation of the overhang of the tool to the height of the shank?

14. What is the objection of having the tool held at an angle in the tool holder when cutting steel?

15. What is the objection to grinding the relief angles of sintered carbide tools on straight wheels?

REFERENCES

Boston, O. W., and Oxford, C. J. "Performance of Cutting Fluids," Transactions, A.S.M.E., MSP-54-2, pp. 9-38, Feb. 15, 1932.

Carboloy Co. "Tool Manual Engineering Bulletin GT-133-R," March, 1943.

Dow Chemical Company. "Machining Magnesium."

Geschelin, J. "Choice of Cutting Fluid Simplified by Research," Automotive Industries, p. 36, July 9, 1932.

Herb, C. O. "Cutting Compounds Used by Ford," Machinery, p. 569, April, 1931.

Johnson, L. B. "Soluble Oil Coolants," Steel, July 7, 1947.

Pure Oil Company. "Cutting Fluids," Product Bulletin No. 61.

Stuart, D. A. Oil Co. "Cutting Fluids for Better Machining."

Vascoloy-Ramet Corp. "Engineering Data VR-400," 1946.

CHAPTER IX

Carbide Forming Tools

Use of the Forming Tool. Forming tools are used extensively in the turret lathe, hand, or automatic screw machine production of such typical parts, an example of which is shown in Fig. 1. Such tools are used especially for sizing and finishing the work. The application and correct design of such tools will be explained in this chapter. While the mathematics in this book has been confined thus far to arithmetic and some algebra, it will be necessary in working with forming tool design to use some trigonometry. However, all the needed formulas for its use are given and special explanations are provided where they appear to be needed.

Flat Forming Tools. Where production quantity is relatively low, the flat type of forming tool as shown in Fig. 2 is satisfactory. Its advantage over the single-point turning tool lies in its ability or capacity to finish the work to the desired size, leaving the difference between the diameters always constant. The tool is sharpened by grinding across the top only. This does not affect its shape or profile but does have the disadvantage of a lowered cutting edge. In order to keep the work to the correct size, therefore, it is necessary to shim the tool after grinding so that its top surface is at the center line of the work. This type of tool also can be used for both planer and shaper work and as a fly cutter in form milling.

Straight Dovetail Forming Tool. However, the tool that is most frequently used for turning the kind of work shown in Fig. 1, is known as the straight dovetail forming tool and is shown in Fig. 3. This tool has the advantage of being easily sharpened by grinding across the top, its shape remaining unchanged. In addition, it is easily adjusted for proper height after it has been resharpened.

These tools are made with an inclination of 10° for front relief or clearance purposes. This inclination remains constant throughout the life of the tool. Since the inclination materially affects the shape of the forming tool, it must be considered both when the tool is designed and when it is constructed.

As shown in Fig. 3, distances X and Y are not equal, X being slightly greater than Y. For 10° of front relief, dimension Y is equal to dimension X multiplied by 0.9848. Stated as a formula,

$$Y = 0.9848 \times X$$

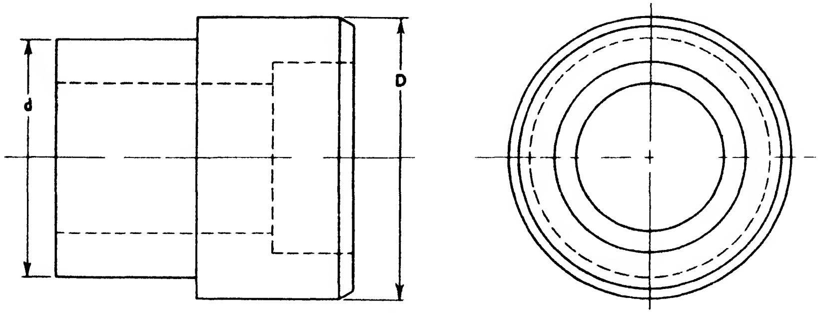


Fig. 1. A Typical Workpiece Such As Would Be Shaped by a Forming Tool

in which 0.9848 is a constant for 10°, being the cosine of the angle. Symbol X represents one-half the distance between the two diameters of the work under consideration and can be taken from the blueprint of the part for which the tool is being designed. Symbol Y is the dimensior to which the tool must be made. It should be noted that if either the 10° relief or the 80° tool angle shown in Fig. 3 is altered after the tool is made, the parts will not be turned out to the required dimensions.

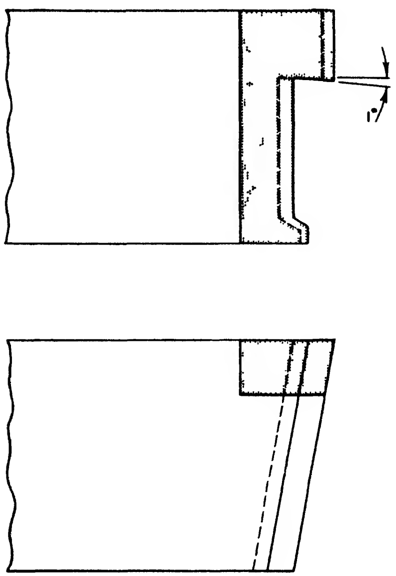


Fig. 2. The Flat Forming Tool

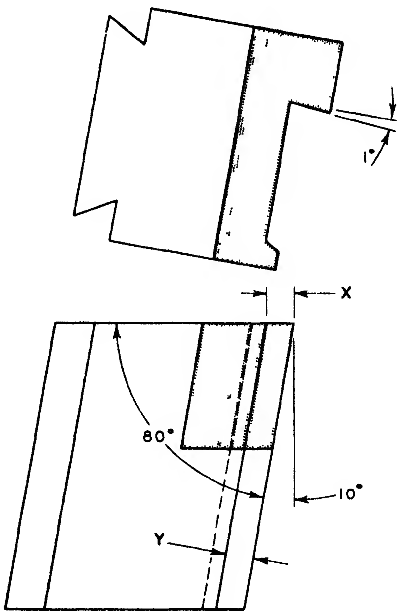


Fig. 3 A Straight Forming Dovetail-Type Tool without Back Rake

As a check on the application of this formula, assume that the two diameters of the part shown in Fig. 1 are 1.500'' and 1.250'' respectively and that it is desired to find the dimensions of X and Y with a 10° front relief and an 80° tool angle.

$$X = \frac{D - d}{2} = \frac{1.500 - 1.250}{2} = 0.125''$$

Substituting this value in the formula given for determining Y, gives

$$Y = .9848 \times X = .9848 \times .125 = .1231''$$

Therefore, the step on the cutting tool, measuring at right angles to the front line, should be .1231''.

Values of constants for different front relief angles can be taken from standard trigonometric tables, since in each case the constant is the cosine of the angle. For the sake of convenience, however, Table I gives several common angles and their constants.

TABLE I. CONSTANTS OF ANGLES

Angle in Degrees	Constant
6	0.9945
6 1/2	0.9936
7	0.9925
7 1/2	0.9914
8	0.9903
8 1/2	0.9890
9	0.9877
9 1/2	0.9863
10	0.9848
10 1/2	0.9832
11	0.9816
12	0.9781

As an additional check on the application of this formula, assume it is desired to find dimension Y for a straight forming tool to turn the part in Fig. 1. Diameter D is 1.750'', d is 1.375''. The clearance is 10° and the lip angle of the tool is 80°.

In this problem, one-half the difference between the two diameters is .1875''. Taking the constant for 10° and substituting the known values in the formula results in

$$Y = .9848 \times .1875 = .1846$$

When turning work involving more than two steps of the type shown

in Fig. 4, the difference between all diameters must be considered in determining the values of Y_1 and Y_2. For example, assume it is de-

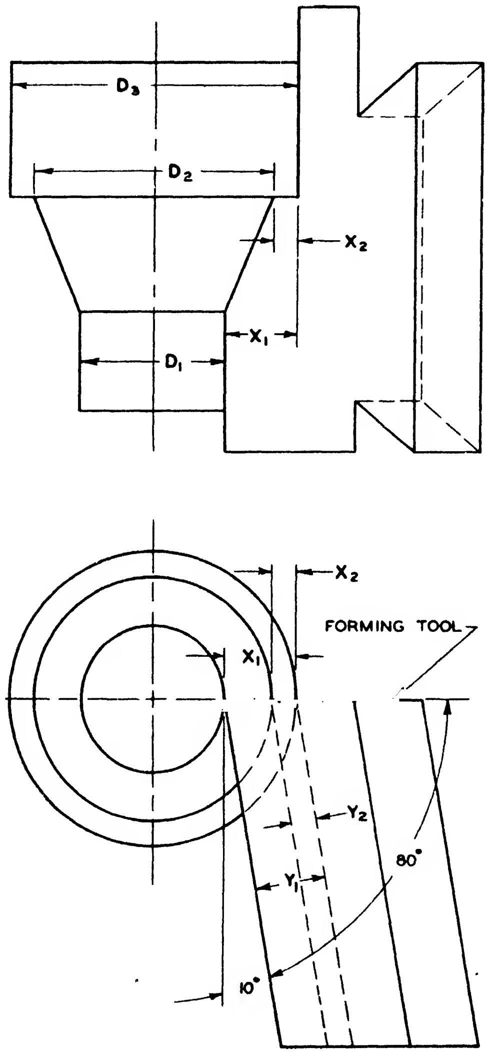


Fig. 4. A Workpiece Having More than Two Steps

sired to compute the dimensions for Y_1 and Y_2 when D_1 = .750", D_2 = 1.125", and D_3 = 1.500". The relief angle is 10° and the lip angle 80° as before.

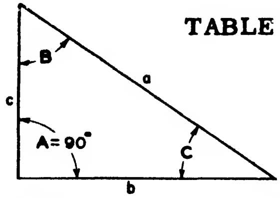


TABLE II. TRIGONOMETRIC FUNCTIONS OF A RIGHT TRIANGLE

No.	To Find Sides	Formulas	
2	a	$\sqrt{b^2 + c^2}$	
3	a	$c \times \text{cosec } C$	$\frac{c}{\sin C}$
4	a	$c \times \sec B$	$\frac{c}{\cos B}$
5	a	$b \times \text{cosec } B$	$\frac{b}{\sin B}$
6	a	$b \times \sec C$	$\frac{b}{\cos C}$
7	b	$\sqrt{a^2 - c^2}$	
8	b	$a \times \sin B$	$\frac{a}{\text{cosec } B}$
9	b	$a \times \cos C$	$\frac{a}{\sec C}$
10	b	$c \times \tan B$	$\frac{c}{\cot B}$
11	b	$c \times \cot C$	$\frac{c}{\tan C}$
12	c	$\sqrt{a^2 - b^2}$	
13	c	$a \times \cos B$	$\frac{a}{\sec B}$
14	c	$a \times \sin C$	$\frac{a}{\cos C}$
15	c	$b \times \tan B$	$\frac{b}{\tan B}$
16	c	$b \times \tan C$	$\frac{b}{\cot C}$

$$X_2 = \frac{D_3 - D_2}{2} = \frac{1.500 - 1.125}{2} = .1875$$

$$Y_2 = .1875 \times .9848 = .1846''$$

$$X_1 = \frac{D_3 - D_1}{2} \quad \frac{1.500 - .750}{2} = .375''$$

$$Y_1 = .375 \times .9848 = .3693$$

Straight Tools with Back Rake. Forming tools with a keen lip angle are preferred by many tool engineers for turning soft materials such as aluminum and magnesium. The relation between dimensions X and Y when the tool has a back rake is shown in an exaggerated manner in Fig. 5. The relationship is quite different from

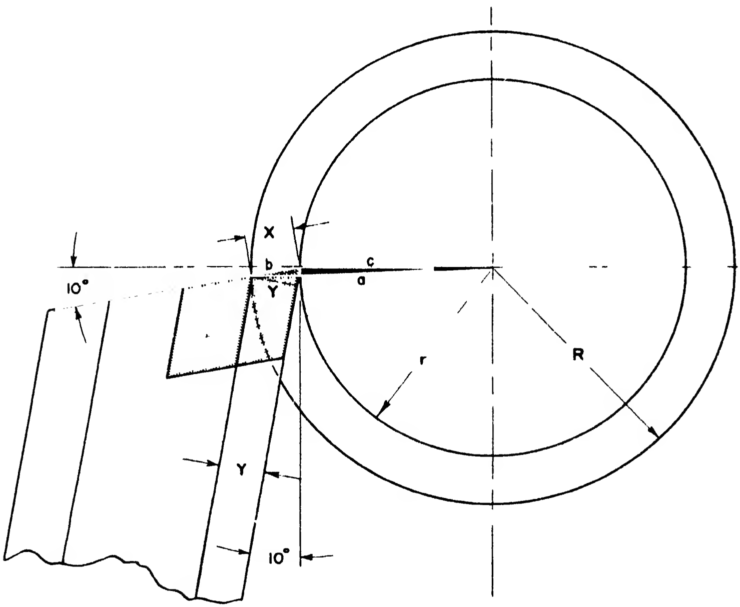


Fig. 5. Forming Tool Having a Keen Lip Angle

that existing in a tool with zero back rake. For a given set of conditions, the problem of the relationship between X and Y can be solved with the aid of the trigonometric formulas pertaining either to right triangles listed in Table II, or to acute triangles as listed in Table III.

Since trigonometry is a subject often unfamiliar to machinists, and easily forgotten unless frequently used, a short review of its terminology is included at this point.

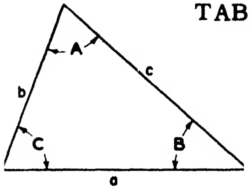


TABLE III. TRIGONOMETRIC FUNCTIONS OF AN OBLIQUE TRIANGLE

No.	To Find	Known	Solution
17	C	A, B	$180° - (A + B)$
18	b	a, B, A	$\frac{a \times \sin B}{\sin A}$
19	c	a, A, C	$\frac{a \times \sin C}{\sin A}$
20	tan A	a, C, b	$\frac{a \times \sin C}{b - (a \times \cos C)}$
21	B	A, C	$180° - (A + C)$
22	sin B	b, A, a	$\frac{b \times \sin A}{a}$
23	A	B, C	$180° - (B + C)$
24	cos A	a, b, c	$\frac{b^2 + c^2 - a^2}{2bc}$
25	sin C	c, A, a	$\frac{c \times \sin A}{a}$
26	cot B	a, C, b	$\frac{a \times \csc C}{b} = \cot C$
27	c	b, C, B	$b \times \sin C \times \csc B$

In trigonometry, the sides of a triangle are usually lettered a, b, and c. This practice differs considerably from that used in geometry, where the corners of the triangle are lettered, and the sides are designated as AB, AC, and BC, and the included angles described as BAC, ACB, and ABC. In trigonometry, the angle opposite the side denoted a is the angle A, or more commonly, is given the Greek letter designation, α (alpha). The angle opposite side b is B or β (beta). The angle opposite side c is C or γ (gamma). Later in this chapter will be problems involving

figures in which the angles have been given Greek letter designations. This practice should cause no difficulty or confusion if it is remembered that the letters merely designate simple angles, or their complements or supplements.

A study of Fig. 5 will show that in order to find the value of X, one must deal with an acute or oblique triangle, having sides a, b, and c equal respectively to R, X, and r. In tool design, the values of R and r may be obtained from the dimensions of the part, while the relief and back rake angles are chosen by the designer.

For the sake of simplicity, the particular oblique triangle called for here is taken out of the problem and illustrated separately in Fig. 6. Angles A, B, and C are shown opposite sides a, b, and c. Assuming that

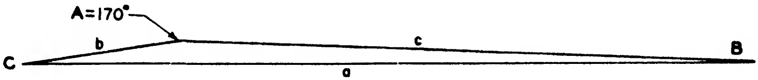


Fig. 6. Oblique Triangle into Which the Problem Is Resolved

the forming tool will have 10° relief and back rake angles, and that the radii of the work will be 1 and 1.250 respectively, the problem is to find angles A, B, C, the side b, which is equal to X, and dimension Y.

The known values are substituted in formula 25 from Table III, resulting in

$$\text{Sin } C = \frac{c \sin A}{a} = \frac{1 \times .17365}{1.250} = .139$$

This is the sine of 7°59' (taken from standard trigonometry tables) which is, therefore, the angle C.

$$\text{Angle } B = 180° - (170° - 7°59') = 2°1'$$

Using formula 18 from Table III, it is found that

$$b = \frac{a \sin B}{\sin A} = \frac{1.250 \times .03519}{.17365} = .2533''$$

Dimension Y can now be found by substituting the known values in formula 8 of Table II. Using the values given in the figure or stated in the problem (a = X = .2533; angle B, the lip angle of the tool, is equal to 90° – 10° – 10° or 70°), gives

$$Y = .2533 \times \sin 70° = .2533 \times .93969 = .238''$$

Standard Dovetail Tools. A dovetail tool of standard design for use on Acme-Gridley automatic machines is shown in Fig. 7. These tools are of typical designs which may or may not have back rake. They

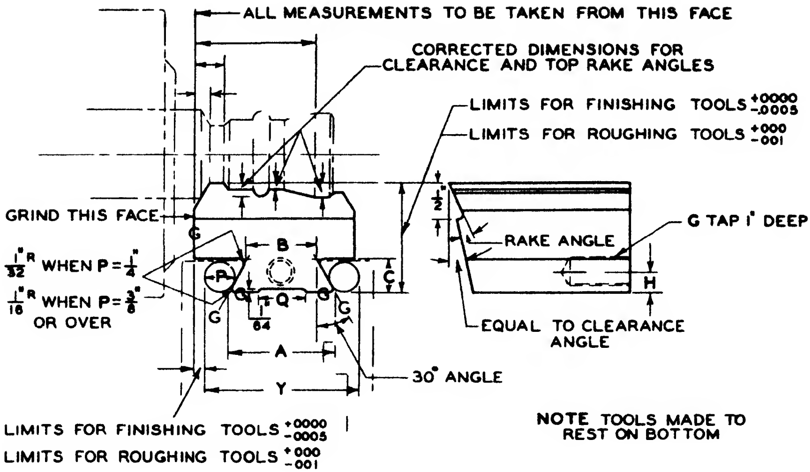


Fig 7 The National Acme-Gridley Standard Dovetail Tool

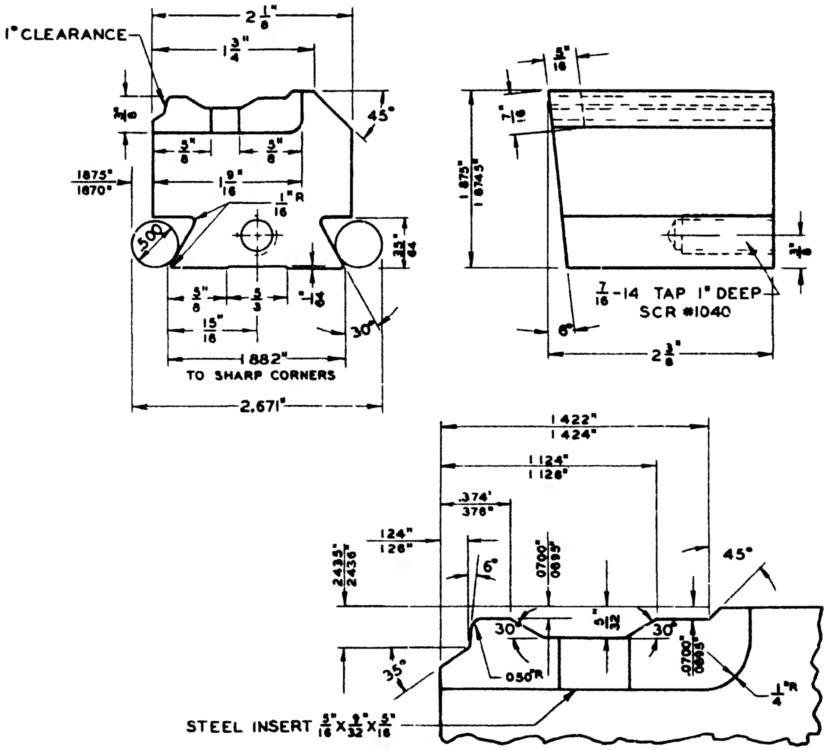


Fig. 8. Design Drawing of a Dovetail Tool Tipped with Carbide

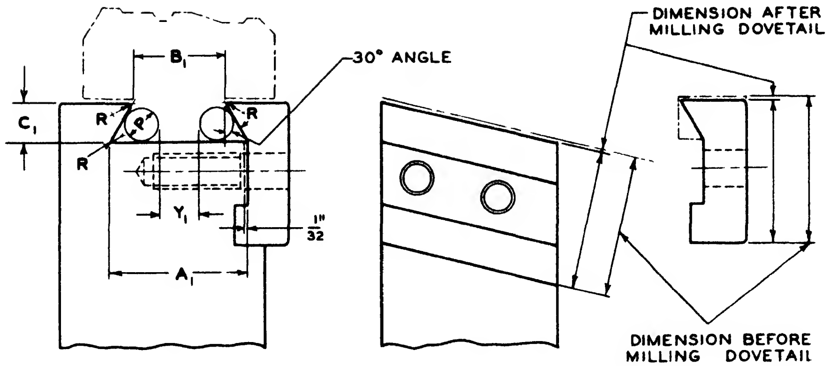


Fig. 9. Standard Dovetail Tool Holder for Acme-Gridley Automatic Machines

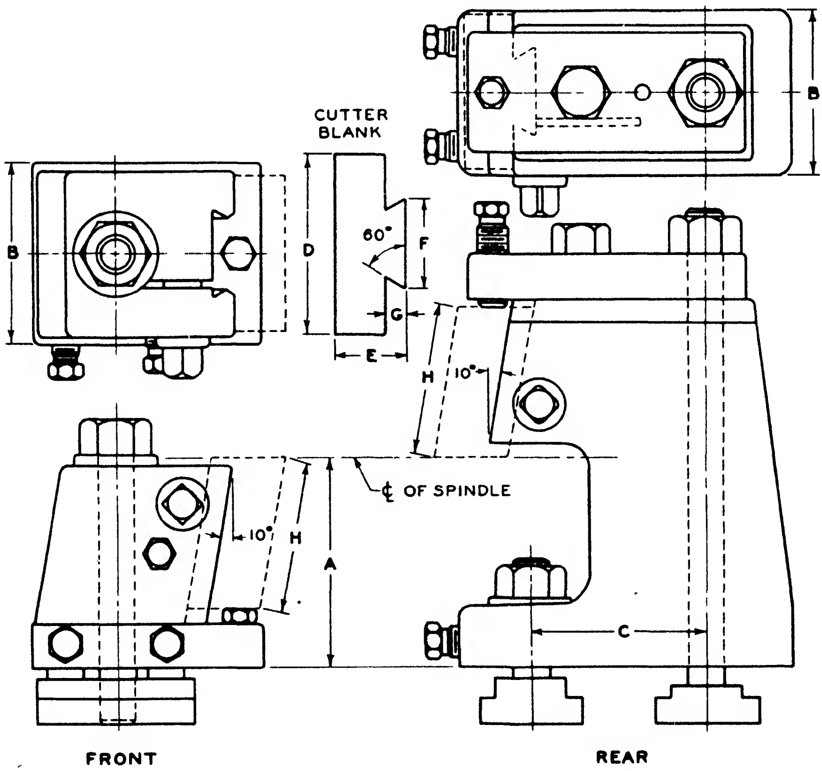


Fig. 10. Standard Dovetail Tool Holders for Warner & Swasey Turret Lathe

can be varied to suit the existing job conditions. Tables IV, V, VI, and VII contain data relating to these tools.

Fig. 8 is an example of a typical design of a standard dovetail forming tool. The design drawing consists of a work layout which includes

TABLE IV. FORMULAS FOR NATIONAL ACME-GRIDLEY STANDARD DOVETAIL TOOLS

$$Y = A + E$$
$$R = B + CF$$
$$B = A - CF$$
$$D = P\left(\cot \frac{90 - \text{angle}}{2}\right) + P$$
$$F = 2 \tan \text{angle}$$
$$E = P\left(\cot \frac{90 + \text{angle}}{2}\right) + P$$

two views of the tool and an enlarged view showing more clearly the profile of the tool. Dimensions are usually given from one reference side only.

Forming-Tool Holders. Figs. 9 and 10 illustrate types of dovetail holders. The tools are clamped from the sides by screws which are visible in the illustrations. The tools can be regulated for height

TABLE V. CONSTANTS TO BE USED IN FORMULAS IN TABLE IV FOR 30° DOVETAILS FOR NATIONAL ACME-GRIDLEY STANDARD TOOLS

Plug "P"	"D"	"F"	"E"
1/4	.6830	1.1547	.3943
5/16	.8537	1.1547	.4929
3/8	1.0245	1.1547	.5915
1/2	1.3660	1.1547	.7886
5/8	1.7075	1.1547	.9858
3/4	2.0490	1.1547	1.1830

in the holder by means of an adjusting screw. These tools are made to fit the machine on which the work is to be done. The size of the machine will, therefore, regulate the size of the holder for the tool. The size of the tool, of course, is dependent on the work for which it is designed. Standard dimensions are given in Tables VIII, IX, X, and XI for tool holders of the type shown in Fig. 9. Table XII gives dimensions for front and rear tool holders of the type shown in Fig. 10.

TABLE VI. DIMENSIONS FOR NATIONAL ACME-GRIDLEY STANDARD DOVETAIL TOOL FORMS

Nominal Size	B	C	A	Y	P	H	G	Screw	Q
5/8	.609	19/64	.951	1.345	1/4	1/2	5/16-18	40	
1	.982	35/64	1.614	2.403	1/2	1/2	5/16-18	40	1/2
1 1/4	1.250	35/64	1.882	2.671	1/2	3/8	7/16-14	1040	5/8
1 5/8	1.606	35/64	2.238	3.026	1/2	3/8	7/16-14	1040	3/8
2 1/8	2.108	43/64	2.883	3.869	5/8	3/8	7/16-14	1040	1

Circular Forming Tools. For turning such work as is shown in Figs. 1 and 4, and for work of a similar nature done in automatic screw machines, circular forming tools such as that shown in Fig. 11 are widely used. These tools are popular because of the ease with which they can be ground or resharpened. Refinishing the tool edge consists merely of grinding the flat face of the tool until practically all of the carbide has been used up.

TABLE VII. DIMENSIONS FOR NATIONAL ACME-GRIDLEY STANDARD SHAVING TOOLS

Nominal Size	B	C	A	Y	P	Q
5/8	.611	17/64	.914	1.308	1/4	
1	.986	17/64	1.289	1.683	1/4	7/16
1 1/4	1.254	17/64	1.557	1.951	1/4	1/2
1 1/2	1.486	17/64	1.789	2.183	1/4	5/8

It will be seen in studying Fig. 11, that the center of the tool is set above the center of the work by the distance h, while the face of the

TABLE VIII. FORMULAS FOR STANDARD DOVETAIL TOOL HOLDERS FOR ACME-GRIDLEY AUTOMATIC MACHINES

$$Y_1 = A_1 - D$$

$$A_1 = B_1 + C_1F$$

$$B_1 = A_1 - C_1F$$

$$D = P\left(\cot \frac{90 - \text{angle}}{2}\right) + P$$

$$F = 2 \tan \times \text{angle}$$

tool is made to set at the center line of the work. This arrangement provides the necessary clearance or relief between the tool and the work which, in the straight dovetail tool, was achieved by inclination of the tool itself. Here, as before, X is the shape or profile of the tool, while Y is the difference between the radii. Y is less than dimension X. In order to get the right shape for the tool dimension, X must be correct.

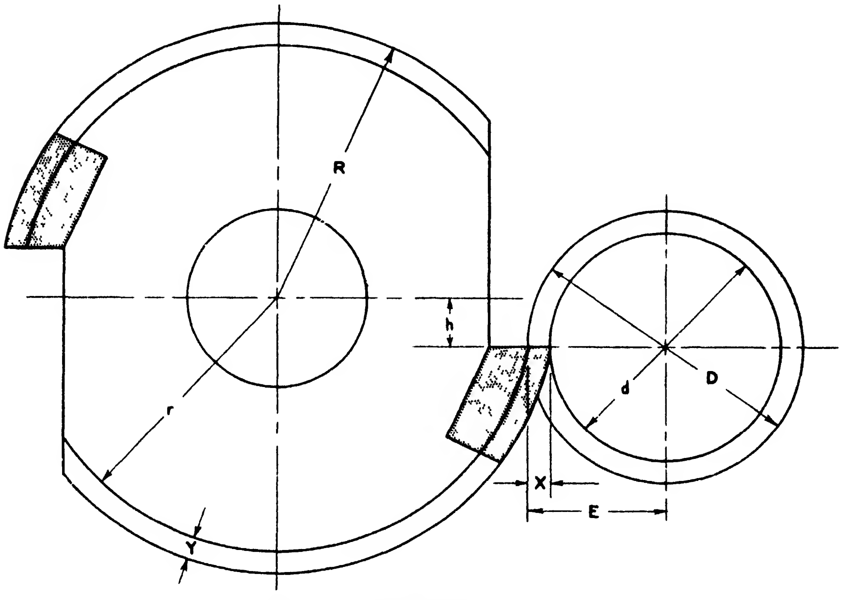


Fig. 11. A Circular Forming Tool

It is necessary, therefore, either to figure out or to lay out the radius, r. Both methods will be explained in detail.

TABLE IX. 30° DOVETAIL CONSTANTS TO BE USED WITH FORMULAS IN TABLE VIII FOR STANDARD TOOL HOLDERS ON ACME-GRIDLEY AUTOMATIC MACHINES

Plug P	D	F
1/4	.683	1.1547
5/16	.8537	1.1547
3/8	1.0245	1.1547
1/2	1.3660	1.1547
5/8	1.7075	1.1547
3/4	2.0490	1.1547

TABLE X. DOVETAIL FORM TOOL HOLDERS FOR ACME-GRIDLEY AUTOMATIC MACHINES

Nominal Size	B_1	C_1	A_1	Y_1	P	Radius R	Radius R_1
5/8	625	.281 .283	.951	.268	1/4	1/32	1/64 maximum
1	1.000	.531 .532	1.614	.589	3/8	1/16	1/64 maximum
1 1/4	1.268	.531 .532	1.882	.857	3/8	1/16	1/64 maximum
1 5/8	1.625	.531 .532	2.238	1.214	3/8	1/16	1/64 maximum
2 1/8	2.125	.656	2.883	1.859	3/8	3/32	1/64 maximum

TABLE XI. HOLDERS FOR SHAVING TOOLS FOR ACME-GRIDLEY AUTOMATIC MACHINES

Nominal Size	B_1	C_1	A_1	Y_1	P	R	R_1
5/8	.625	.250 .249	.913	.230	1/4	1/32	1/64 maximum
1	1.000	.250 .249	1.288	.605	1/4	1/32	1/64 maximum
1 1/4	1.268	.250 .249	1.556	.873	1/4	1/32	1/64 maximum
1 1/2	1.500	.250 .249	1.788	1.105	1/4	1/32	1/64 maximum

Mathematical Determination of Tool Radii. The following two formulas may be used to determine the radii of circular forming tools which may be employed for turning work to desired dimensions:

$$r = \sqrt{\left(\sqrt{R^2 - h^2} - X\right)^2 + h^2}$$

$$R = \sqrt{\left(\sqrt{r^2 - h^2} + X\right)^2 + h^2}$$

In these two formulas, r is the required radius of the forming tool

TABLE XII. DIMENSIONS OF DOVETAIL TOOL HOLDERS FOR WARNER & SWASEY TURRET LATHES

A	B	C	D Max.	E Max.	F	G	H
3 1/2	3	3	3	1 1/4	1 1/2	3/8	2 1/2
4 1/4	2 7/8 (front) 2 9/16 (rear)	3	3	1 1/4	1 1/2	3/8	3 1/4
4 3/4	3	3	3	1 1/4	1 1/2	3/8	3
4 5/8	3	3 1/4	3	1 1/4	1 1/2	3/8	3
5 1/8	3	3 1/2	3	1 1/4	1 1/2	3/8	3

in inches: R is the largest radius of the forming tool in inches; X is the difference in radii, or steps of the work, in inches; and h is the amount the center lines of the tool and the work are offset.

As a check on the application of these formulas, assume the work shown in Fig. 1 has two diameters: 1.5" and 1.25". To form the work, the larger radius of the tool is to be 1.5", and the offset of the centers is to be .250". What should be the dimension of the small radius if the tool does not have a back rake?

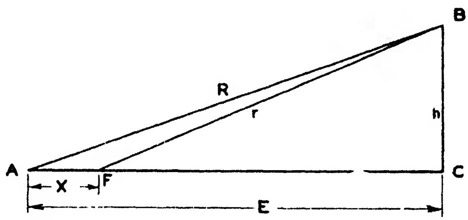


Fig. 12. The Method for Graphic Solution of Design

Since R = 1.5", h = .250", and $X = \frac{1.5 - 1.25}{2} = .125$", the value of r is found by substituting the known values in the formula, resulting in

$$r = \sqrt{\left(\sqrt{1.5^2 - 0.25^2} - .125\right)^2 + .25^2}$$

$$r = \sqrt{1.896}$$

$$r = 1.377$$

Were the center line of the tool and the work on the same level, the radii of the tool would have been 1.5" and 1.375" respectively. Since the forming tool is above the center line of the work, the radius R = 1.5", and r = 1.377".

Graphic Solution of Tool Radii. When the accuracy of the work is essential, the computation method of determining the radii, as just presented, is the best. However, there are many cases where accuracy is not as important and the problem can then be solved graph-

ically. Such a system is shown in Fig. 12. The method of procedure is as follows:

Draw the base line, AC. Erect a perpendicular at C and take a length equal to h, the desired center line offset. With B as the center and R as the radius, strike an arc, B – A, from point B, cutting the base line at A. Lay off distance X on the base line as in the figure, obtaining point F. Connect point F with B, obtaining radius r.

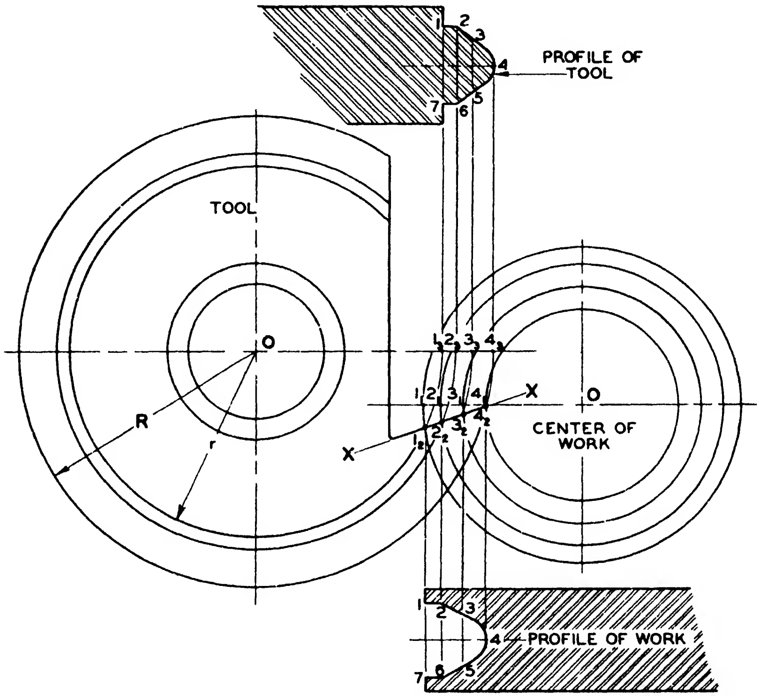


Fig. 13. Layout for a Circular Forming Tool with Positive Back Rake

When the layout is done with care, the error in the solution is slight. When greater accuracy is required, the layout may be done to an enlarged scale, 4, 5, or even 10 to 1. The error in such an increased scale will be negligible.

Graphic Determination of Profile. The profile or shape of a circular forming tool having a positive back rake angle also can be arrived at graphically when the accuracy of the work will permit the use of such a method. By accurate construction to a much enlarged scale, it is quite possible to get tool dimensions within the limit of plus and minus .001" or less. For this purpose, the system shown in Fig. 13 can be used. This illustration shows a forming tool with positive back rake, the work, and the profile of the work.

The graphic determination of profile is accomplished as follows: points 1 to 7, or as many points as may be necessary, are marked on the profile of the work. These are projected on the center line of the

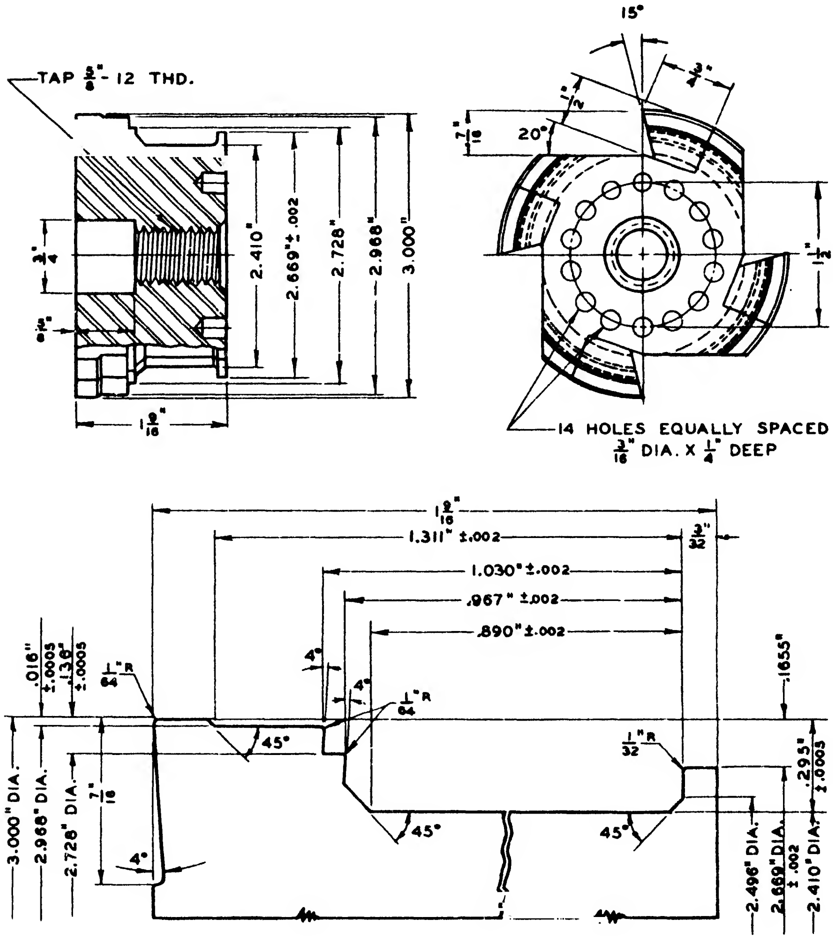


Fig. 14. Typical Circular Forming Tool Design

work and appear as points 1_1, 2_2, 3_3, and so on. From the center line of the work the points are transferred to line X – X by arcs drawn from the center of the work piece 0, producing intersections 1_2, 2_2, 3_2, and so on. Circles are drawn next from the center of the tool, 0, through points 1_2, 2_2, 3_2, etc., intersecting the center line of the tool at points 1_3, 2_3, 3_3, and so on. These are the desired points that will appear on the center line of the tool profile and should be projected there. The widths

of the tool profile will be 1 – 7, 2 – 6, 3 - 5, and 4. Joining these points produces the profile of the tool.

A typical design for a circular forming tool is illustrated in Fig. 14. This drawing gives all the dimensions required as well as an enlarged view of the sintered carbide tip.

Computation of Radii. When the accuracy demanded in the tool is such that a graphic solution will not suffice, a formula can be

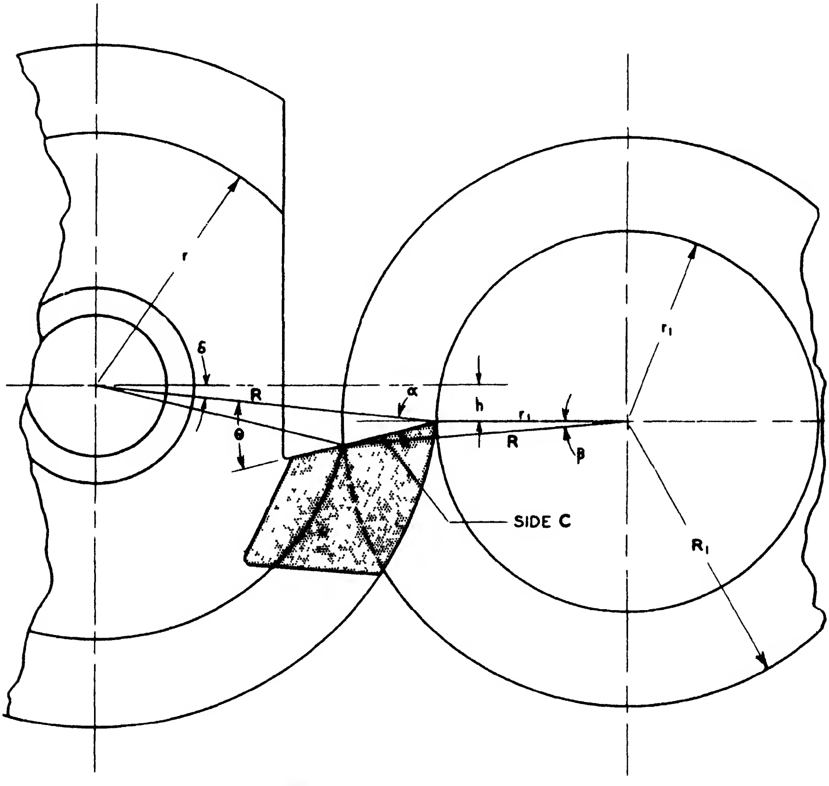


Fig. 15. Geometry of the Circular Forming Tool

used for computing the small radius, or radii, if the contour of the tool is such that the tool has more than one small radius. In the computation, it is assumed that the point of the tool is set at the center of the work as shown in Fig. 15. Then

$$r = \sqrt{C^2 + R^2 - 2\ CR \cos\theta}$$

in which

r = the small radius of the tool in inches

R = the large radius of the tool in inches

$C = \dfrac{R_1 \sin \theta}{\sin a}$, where C is the face of the forming tool as shown in the figure. The formula for C is #18 taken from Table III.

Angle β = rake angle a – angle ϕ (phi) (from simple geometry)

Sine $\phi = \dfrac{r_1 \sin a}{R_1}$ (from formula 22, Table III)

r_1 = the small radius of the work in inches

R_1 = the large radius of the work in inches

$\theta = \delta$ (delta) $- a$; or sine $\delta = \dfrac{h}{R}$

To illustrate the method of computation, assume it is desired to find the small radius of the tool when the values of the dimensions in Fig. 15 are as follows: $R_1 = 1.5''$, $r_1 = 1''$, $R = 1.8''$, the rake angle = 10°, and $h = 0.2''$. The problem cannot be attacked directly because it is necessary to find first, the angles ϕ, δ, and θ. From the geometry of the figure and the formulas cited, it is known that

$$\text{Sin } \phi = \frac{r_1 \sin a}{R_1} = \frac{1 \times \sin 10^\circ}{1.5} = \frac{1 \times .174}{1.5} = .116$$

Then

$$\phi = 6^\circ 40', \text{ and } \beta = a - \phi = 10^\circ - (6^\circ 40') = 3^\circ 20'$$

$$C = \frac{R_1 \sin \beta}{\sin a} = \frac{1.5 \times .058}{.174} = .500''$$

$$\text{Sin } \delta = \frac{h}{R} = \frac{.2}{.18} = .111$$

Hence, the angle $\delta = 6^\circ 23'$ and angle $\theta = 10^\circ + 6^\circ 23'$ and angle $\theta = 10^\circ + 6^\circ 23' = 16^\circ 23'$.

These values are substituted in the formula given for the computation of radii, resulting in

$$r = \sqrt{.5^2 + 1.8^2 - 2 \times .5 \times 1.8 \times \cos \theta}, \text{ or}$$

$$r = \sqrt{.25 + 3.24 - 2 \times .5 \times 1.8 \times .959} = \sqrt{1.77} = 1.330''$$

Should the contour of the work require additional small radii, they will all have to be determined separately for mathematically accurate dimensions. Otherwise, whenever the degree of allowable inaccuracy permits, the solution of the problem should be achieved graphically.

TABLE XIII. SIZE OF CIRCULAR FORMING TOOLS AND OFFSET, h, FOR BROWNE & SHARPE, NATIONAL ACME AND CLEVELAND AUTOMATIC SCREW MACHINES

Machine	Size of Machine	Radius, R, in Inches	Offset, h, in Inches
Browne & Sharpe	00	0.875	.125
	0	1.125	.156
	2	1.500	.250
	6	2.000	.312
National Acme	51	0.75	.937
	52	1.000	.937
	53	1.875	.125
	54	1.250	.156
	55	1.250	.156
	56	1.50	.187
Cleveland	1/4	0.625	.0312
	3/8	0.84375	.0625
	5/8	1.156	.0625
	7/8	1.187	.0625
	1 1/4	1.375	.0625
	2	1.375	.0625
	2 1/4	1.625	.125
	2 2/4	1.875	.156
	3 1/4	1.875	.156
	4 1/4	2.500	.250

TABLE XIV. STANDARD DIMENSIONS FOR CIRCULAR FORMING-TOOL HOLDERS

A	B	C	D	E Max.	E Min.	F Max.	F Min.	G	H	Thread J
2 1/8	3/4	2	1 3/4	2 1/8	2 1/8	1	3/8	3/4	1/8	1/2-13P
3	3/4	2 1/2	2 1/4	2 1/8	2 1/8	1	3/8	3/4	1/8	1/2-13P
3 1/2	1	3	3	3	2 1/2	2	1/2	1	3/16	5/8-11P
4 1/4	1	3	3	3	2 1/2	2	1/2	1	3/16	5/8-11P
4 5/8	1	3	3 1/2	3	2 1/2	2	1/2	1	3/16	5/8-11P
4 3/4	1	3	3	3	2 1/2	2	1/2	1	3/16	5/8-11P
5 1/8	1	3	3 1/2	3	2 1/2	2	1/2	1	3/16	5/8-11P

The outside diameters of circular tools used in automatic screw machines are pretty well standardized. These dimensions are given in Table XIII and may be used as a guide for designing purposes.

Circular Forming-Tool Holders. Fig. 16 illustrates how circular tools may be mounted in toolholders for use in turret lathes or automatic screw machines. As shown in the drawing, the tool can be adjusted for the position of its cutting edge by a screw, operating in a slot. This permits rocking of the tool. The right-hand view shows a

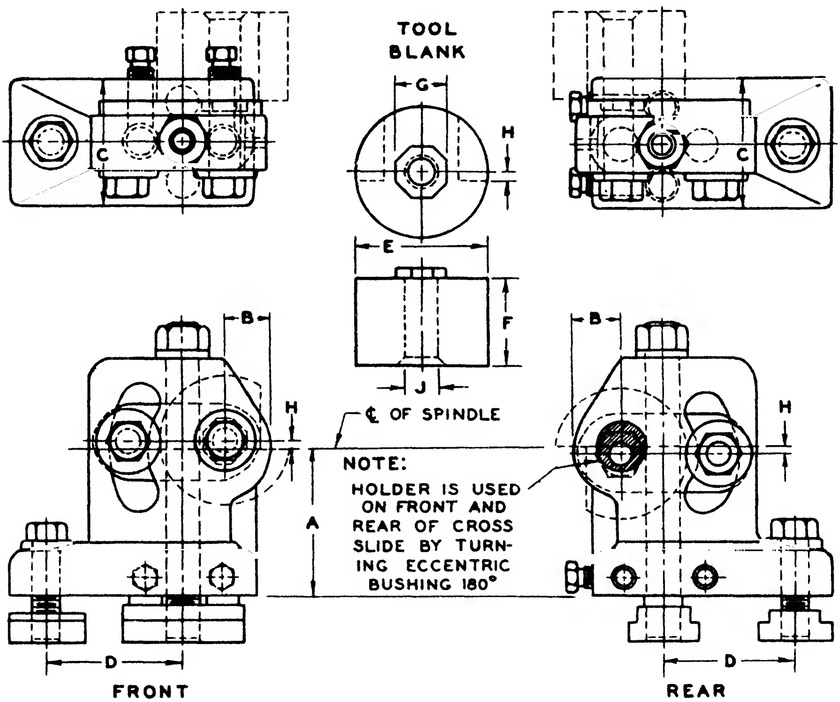


Fig. 16. Circular Forming-Tool Holder

tool used in the rear of the work with the toolholder held in position on the cross slide by means of bolts. Standard dimensions for circular forming-tool holders are given in Table XIV.

Cutting Speeds and Feeds. Sintered carbide forming tools are to be operated at much the same speeds as recommended for single-point cutting tools. Too low a speed will cause an excessive built-up edge on the tool, resulting in poor finish on the work, excessive edge wear, and a glazed work surface.

The feed for forming tools is usually low, but it is best to feed in at least .001" per revolution of the work. Too low a feed will cause ex-

cessive wear and premature failure of the cutting edge. Feeds which are too heavy should not be attempted because forming tools are finishing tools and are not intended to remove large amounts of material.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Define the forming tool and state its purpose.
2. What is the essential difference between the straight forming tool and the circular forming tool?
3. When is the straight forming tool used?
4. For what type of work is the circular forming tool used?
5. Why is it necessary to make correction for the form of the forming tool?
6. A dovetail forming tool is to be used without back rake. The front clearance is to be 6°. The diameters of the work are to be 2'' and 1.5'' Determine the perpendicular distance between the flanks of the tool, designated by Y in Fig. 3.
7. If, in the preceding problem, the front relief is to be 10°, what should be distance Y?
8. If the tool in problem 6 is to have a 10° back rake in addition to the front relief of 6°, determine the distance or dimension Y shown in Fig. 5. (Suggestion: use formulas 25 and 18 from Table III)
9. If a circular tool without back rake is to be designed for the work in problem 6 and its large radius is 1.5'', what should be the small radius of the tool if h is 1/8''?
10. If the tool in problem 9 is to have a 10° back rake, determine graphically its small radius.

REFERENCES

Cole, C. B. Tool Design. Chicago: American Technical Society, 1941.

Donaldson, C., and LeCain, G. H. Tool Design. New York: Harper and Brothers, 1943.

Kurrein, M., and Lea, F. C. Cutting Tools for Metal Machining. Chicago: J. B. Lippincott Co., 1940.

Kozacka, J. S. "Precision Tools and Inspection," Science and Mechanics Magazine, October, 1944.

National Acme Company, "Handbook for Operators of Acme-Gridley Multiple Spindle Machines."

Warner & Swasey Company, "Turret Lathe Tools."

CHAPTER X

Carbide Milling Cutters

Description of Milling. Milling is a process in which rotary cutters, provided with hard and sharpened teeth, are used to cut metal. Milling differs from other operations, such as turning, in that flat, angular, or formed surfaces are usually produced. The cut of each milling cutter tooth, in addition, is not continuous but intermittent, making one cut at each revolution. This feature is illustrated in Fig. 1. In general, the milling machine table holds the workpiece and advances or feeds it slowly under the rotating cutter in such a way that a surface cut is completed in one advancing motion of the work piece. Fig. 2 illustrates a typical setup in a milling machine for facing.

Milling machines differ widely in construction. Some are designed for tool work or for jobbing shop applications, while others are made with special fixtures to hold the work and have special cutters to machine it. Milling has largely superseded shaping and planing for repetitive work and large quantity production because it removes metal much faster and produces surfaces with a better finish.

Milling Cutters. Milling cutters are revolving tools having one or several cutting edges of identical form, equally spaced on the circumference of the cutter. The cutting action, or the removal of the material by a milling tooth, is governed by the same basic laws, so far as the separation of the chip from the parent metal is concerned, as in turning, shaping, or planing. It is in the revolution of the tool itself that the action differs from that in turning. Also, in turning, the action is usually continuous, while in milling, the action is intermittent. The tooth of the milling cutter comes in contact with the work material only for a short time during the period of one revolution. The rest of the time it turns through the air. Again, in turning operations, the chips are of uniform thickness. In milling they are short and of varied thickness. This thickness of chip is illustrated graphically in Fig. 3. The action caused by the engaging and disengaging of the teeth in the work results in a "hammering" of the teeth on the work which is in itself detrimental to the tool and must be taken into consideration during the design and use of the tool.

Nomenclature. For a discussion of the details of carbide-tipped cutters, it is necessary first to establish a nomenclature. This will prevent confusion and misunderstanding regarding the terms used. As

an aid in defining these terms, Figs. 4 and 5 have been prepared so as to include the various points as they are discussed. A plain milling cutter, tipped with sintered carbide, is shown in Fig. 4. Fig. 5 illus-

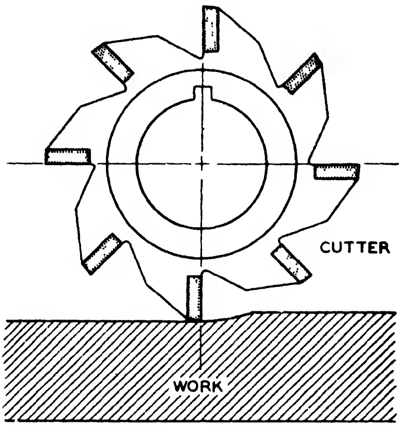


Fig. 1. The Cutting Action of the Milling Cutter

Fig. 2. Horizontal Milling Machine Setup for Face Milling
Courtesy of Brown & Sharpe Mfg Co.

trates an inserted-blade milling cutter together with an enlarged view of the blade.

Tooth Face. The face of the tooth is that part of the tool which fronts against the work.

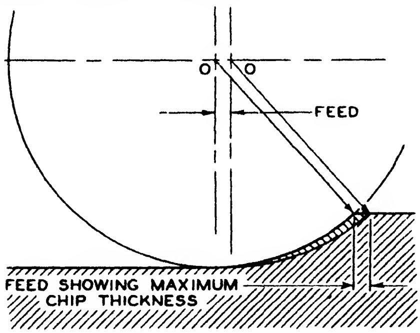


Fig. 3. The Varying Thickness of the Milling Chip

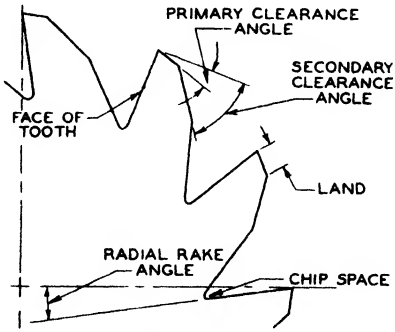


Fig. 4. Nomenclature of the Plain Milling Cutter

Radial Rake Angle. The radial rake angle is the side rake angle of the single-point tool (discussed in Chapter III) translated into rotary terms. It has a great influence on the cutting action of any milling cutter and its form and function should be studied closely. This angle

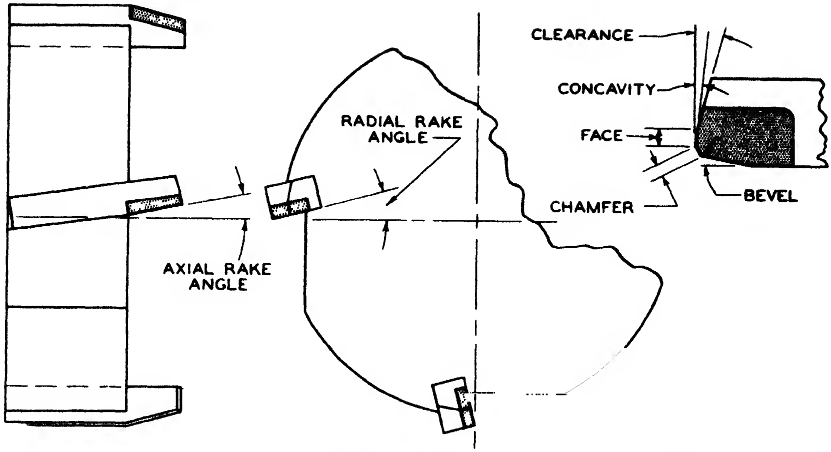


Fig. 5. Nomenclature of the Inserted Blade (or Tooth) Milling Cutter

has been the subject of much research by many investigators with the result that under today's high-speed production using milling cutters tipped with carbide, it usually is set between 5° and 10° negative. However, some cutters are still made with positive radial rake for various purposes. Negative rake requires considerably more power to operate than positive rake.

Axial Rake Angle. This angle refers to the direction in which the teeth are set on the periphery of the cutter. The angle may be either positive or negative. The negative angle, illustrated in Fig. 6, is commonly used on milling cutters designed for general shop work on steel and those tipped with the hard but brittle sintered carbides. This setting of the cutter blades protects the chamfer or radius of the tooth point when work is done, because the cut is started away from the cutting edge. In other words, that part of the tooth behind the cutting point engages the work before the point, thus absorbing the impact force with the body of the tooth and at the same time protecting the point.

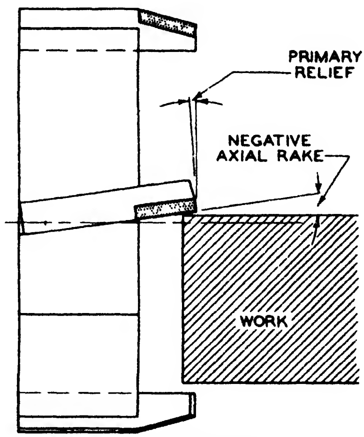


Fig. 6. A Cutter Having a Negative Axial Rake

The axial angle in milling cutters is the equivalent of the back rake of the single-point tool described in Chapter III. It will be remembered that for interrupted cuts in turning, facing, and boring, a negative back rake angle of from 5° to 45° was recommended. At that time it was

pointed out that a negative rake tool first engages the work behind the point, thus protecting it from possible damage. This negative axial rake is shown in Fig. 6.

Axial rake angles in milling cutters tipped with sintered carbides usually vary between 10° and 20° positive for machining aluminum and magnesium, and between 5° and 15° negative for machining steel of more than 180 Brinell hardness. These features of design will be discussed subsequently in sections of the book which are devoted to the design of milling cutters and other multiple-point cutting tools.

Land. The land is that portion of the tooth which is just behind the cutting edge, as shown in Fig. 4. The land is small on a new cutter, but increases with each regrinding. It varies between 1/32'' for small cutters to 3/32'' for larger ones.

Primary Relief. The primary relief is the first relief behind the cutting edge. Its purpose is to prevent the cutter from dragging on the work. The amount of this clearance or relief is small, usually from 3° to 6°. The primary clearance extends over the length of the land.

Secondary Relief. Behind the primary relief, there is a secondary clearance angle. Its function is to provide adequate chip clearance or space, and to permit the removal of a small amount of material during sharpening. These two reliefs are similar in purpose to the primary and secondary front reliefs discussed in Chapter III regarding single-point cutting tools.

Bevel Angle. The bevel angle on the face mill serves the same purpose as the side cutting-edge angle in the single-point tool. Its function is to permit the cutter to begin cutting higher on the tool than the chamfer, thus protecting the vulnerable point and increasing tool and cutter life.

In most milling cutters, a bevel angle of 15° is satisfactory, but in many cases it may be necessary to have this angle be as much as 35° Bevel angles as great as 45° are sometimes designed for tools. The length of the angle should be sufficient to cover the entire depth of the cut. In applications where the bevel angle must be zero, the radial rake should be negative from 5° to 10°.

Chamfer. All milling cutters of the type used for facing should have the corners beveled. This beveling removes the sharp point which tends to break and crumble. The chamfer may be from 1/16'' by 45° to 1/8'' by 45°. When a radius grinding attachment is available in the tool grinding room, a suitable corner radius in place of a bevel chamfer will be more satisfactory.

Concavity of the Tooth Face. The concavity of the tooth face of the facing cutter, as shown in Fig. 5, is small, being ordinarily from 1/2° to 1°, but sometimes having a maximum of 2°. Its function is to prevent the face of the tooth from rubbing on the work. This would cause heating of the work and unnecessary dulling of the cutter. The secondary clearance toward the heel is usually from 5° to 7°. The width of the face need not be greater than 1/8'' to 3/16''. The rest of the tooth depth

constitutes the heel which is ground 5° to 7° for the secondary tooth clearance.

Making the Milling Cutter. The sintered-carbide milling cutter consists of a body which has cutting teeth fixed to it by one method or another. The body may be made of high-grade steel either cast or forged, of cast iron, or it may be machined from bar steel or plate. It is often made of special alloys. Usually, the body is of a

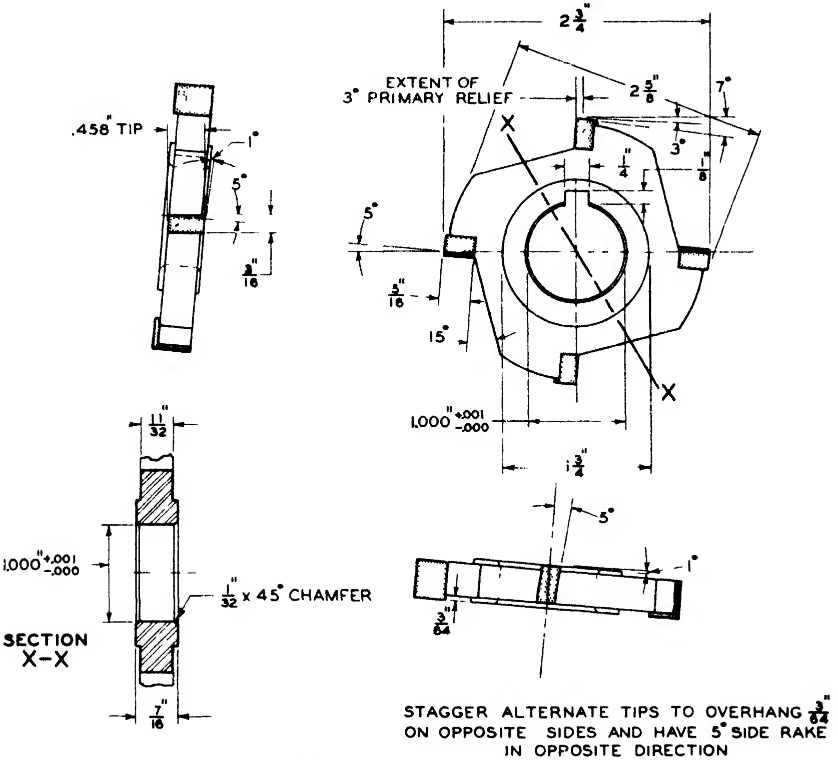


Fig. 7. A Stagger-Tooth, Slot Milling Cutter

medium-grade machine steel containing about 0.45 per cent carbon. The carbide teeth are either brazed to the body or are held in place by means of screws and wedges. Frequently, the inserted blade is tipped with sintered carbide and then held in the body of the cutter by mechanical means.

Shown in Fig. 7 is a milling cutter with sintered carbide teeth. It will be noted that the teeth are staggered. That is, one tooth extends to one side and the next extends to the opposite side. The body of the cutter is made of steel to which the carbide teeth have been brazed, then

ground to the desired width, diameter, and relief or clearance angles. This particular cutter has negative radial and negative axial rake angles of 5° each, a common practice with many tool designers.

There are three steps in making a milling cutter of the type shown in Fig. 7. These are the shaping of the cutter body and the cutting of the recess for the carbide blank, the brazing on of the carbide tips, and the grinding of the assembled tips. Frequently, it is necessary to machine the arbor hole.

The rough machining may be done on a lathe or a turret lathe and finished in an engine lathe. The keyway is cut on a keyseating machine or a broaching machine. The chip clearances and carbide tip seats are next machined, usually in a milling machine. Following this, the cutters are ready for brazing.

When a large number of cutters are to be made, the bodies may be forged to nearly finished dimensions. Only finishing operations are then performed on the body.

Milling cutter bodies for any type of cutter should be carefully prepared for brazing. Otherwise, unsatisfactory brazes may result. The preparation for brazing consists of degreasing the cutter body in some suitable solution, followed by a careful grit blasting of all surfaces that might contain scale or oxides. The same procedure should be followed in preparing the carbide tips. Even the brazing material should be free from grease or oil and should not be handled with the hands lest it become coated with grease or moisture.

When only a small number of cutter bodies and tips are to be assembled, the cleaning or degreasing may be done by dipping in hydrochloric acid, followed by a bath in carbon tetrachloride. This procedure will remove thin films of grease or oil. After this operation, care should be exercised that the parts are not allowed to collect dirt or grease before the actual brazing.

The brazing of sintered carbide tips to milling cutter teeth may be done with the oxyacetylene torch, in a furnace, or by heating with an induction coil as was described in Chapter IV. When only a few cutters are to be brazed, the oxyacetylene torch method of brazing is satisfactory, and is preferred by many toolmakers. When larger quantities of cutters are to be made, the furnace or induction methods are to be preferred because they are so much faster and lend themselves to better control.

Brazing by the torch method is illustrated in Fig. 8. The cutter body is carefully prepared as previously described, and the recess coated with a thin film of flux. A piece of silver solder is placed in the recess and this is then coated with flux. Finally, the carbide tip is coated with flux and placed in the recess or seat.

The whole assembly is held firmly in a vise or special fixture and the heat applied to it from the back of the tooth. Heating should be done slowly and uniformly, moving the torch from underneath to the sides and finally to the tip or, in other words, rotating the torch until the silver

solder has melted. At this point, the torch should be moved slightly away from the tooth and the carbide tip "puddled" or wiped in place with a poker, as shown in Fig. 8. The torch is then taken away and the tip held in place with the poker until the silver solder completely solidifies. The cutter is then rotated to the next tooth and the procedure repeated.

The reason for puddling is twofold. First, the excess flux and brazing material must be squeezed out from under the tip. Second, any air or gas that may have been trapped must be expelled. Either of these conditions will cause a poor braze. In torch brazing, the whole assembly in the area where heat is to be applied should be covered with a copious

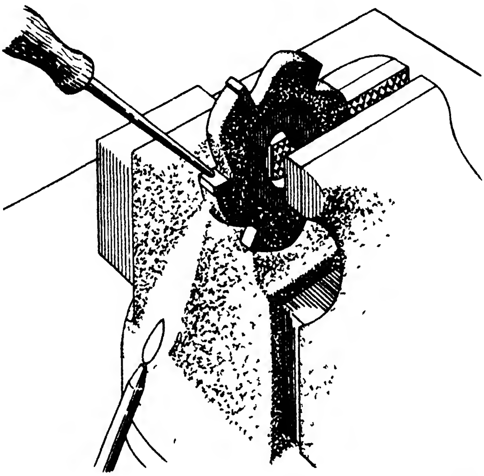


Fig. 8. Brazing the Tip in Place with an Oxyacetylene Torch

amount of flux so that the surface of work will not oxidize or scale.

The flame used should be neutral or slightly reducing. That is, it should have more gas present than usual. This will prevent oxidation. A nonoxidizing flame is shown in Fig. 6 of Chapter IV.

In holding the cutter in a vise, care should be taken against clamping it too tight, since there is danger of cracking the tips previously brazed in. This accident may be prevented by lining the jaws of the vise with some soft material such as copper.

The cutter body, the tip blank, and the brazing material are prepared in the same way for furnace brazing as for torch brazing. However, it is necessary in furnace brazing to tie the tips in position with carbon rods and nichrome wire or asbestos string. This procedure is shown in Fig. 9. The assembly is then placed in a furnace which is operating with a neutral or slightly reducing atmosphere, and at a temperature from 100° to 200° F. above the melting point of the brazing material. The method of tying the tips in place can also be used in con-

junction with torch brazing. However, when this is done, a small torch and flame must be used.

Another method of brazing carbide tips to milling cutter bodies is that in which a copper-tube induction heating coil is used. The process is similar to that described in connection with the single-point tools and a small sketch is shown in Fig. 10. In this arrangement, the heating is fast and is concentrated at the tip and the adjacent area of the cutter body. Each tooth is brazed separately into position. Brazing one tooth in at a time has the advantage of allowing each tooth to be puddled or wiped into position by the operator, thus assuring a sound and strong union of the tip to the cutter body. When properly brazed, the tip will

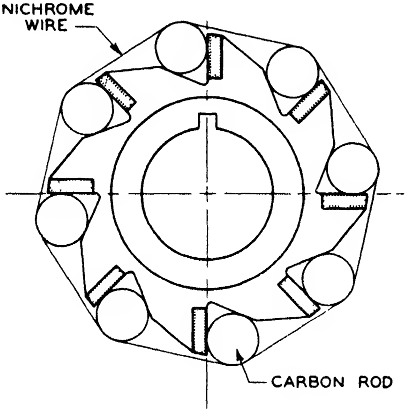


Fig. 9. Use of Carbon Rods To Hold Carbide Tips in Place During Furnace Brazing

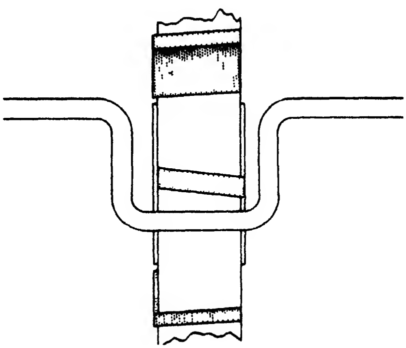


Fig. 10. Brazing of Tips to Cutter Bodies by Means of An Induction Coil

not easily come off under the cutting action of the tool, for the braze, in the direction of stress, is actually much stronger than either the cutter body material or the carbide tip.

Inserted Carbide Blades. Larger milling cutters of the facing variety have blades inserted into the body of the cutter. These teeth usually are tipped with carbide. The brazing of the tips to the blades is done in the same manner as was described for the brazing of single-point tools in Chapter IV. In this case, the milling cutter actually consists of a number of single-point tools mounted on the body. An example of this form of tool is shown in Fig. 5.

In all methods of brazing, the primary requisite is extreme cleanliness of the cutter body, the carbide tip, and the brazing material. The flux may remove a certain amount of the impurities and even a thin film of grease from the surfaces to be brazed. However, to insure maximum bond, the work should be thoroughly cleaned by either chemical or mechanical means before the flux is applied.

Special Brazing Procedures. Other multiple-point cutting

tools such as reamers, counterbores, countersinks, or combinations of them in all manner of designs, can be brazed by any one of the methods described here and in Chapter IV. However, some tools can be brazed by one method more readily than by another.

Most brazing jobs can be done with the oxyacetylene torch except when the teeth are spaced so closely together that there is danger of overheating the adjacent teeth or flutes previously brazed, thus destroying the joint.

Typical of the special brazing procedures is that in which the carbide tips are fastened to the helical flutes of the four flute reamer. In carbide tipped reamers, the tips are not long, usually being from 1/4'' in small tools to as much as 1 1/2'' in the largest sizes. The reamer is best brazed in the furnace or by the use of the induction coil. For brazing by either the induction coil or in the controlled atmosphere furnace. the reamer is prepared by assembling the body and the carbide blanks with the brazing material, usually Easy-Flo #3, the whole being held together and in position by graphite rods and nichrome wire. This setup is shown in Fig. 11.

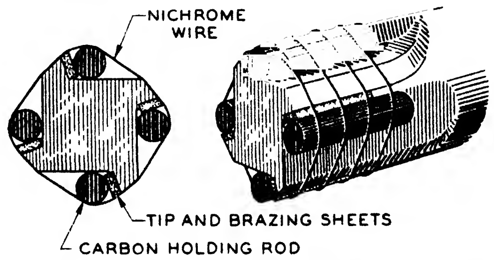


Fig. 11. Assembly of a Four-Flute Reamer for Furnace Brazing of Tips

In (A) of Fig. 12 is shown the setup for brazing a reamer by means of heating in a single-turn induction coil. The heating is slow in this process but permits manipulation or ''wetting in'' of the surfaces to be brazed by moving the tip with a suitable poking tool. The disadvantage is the slowness of the heating cycle, but where accuracy and soundness of brazes are of first importance, this method will produce excellent results. In (B) of Fig. 12 is pictured the reamer setup for brazing in a multiple-turn induction coil. The heating cycle here is much faster than in the single-turn coil. The results of this method of brazing are good since the heating cycle is faster, allowing less chance for oxidation of the shank and the brazing material.

Brazing Special Milling Cutters. Milling cutters of the type shown in Fig. 13 have the body made of steel; of Meehanite, a special alloy; or of a combination of the two. The cutting points are made of carbide brazed to the recesses. This particular cutter is a stagger-tooth slot milling tool similar to that shown in Fig. 7.

In the preparation of such cutter bodies, and of all cutter bodies to a somewhat lesser degree, the recesses should be absolutely flat. Any unevenness or rounding of the surfaces will contribute to the cracking of the cutter teeth while in operation. Before any brazing is attempted, and especially after the cutter has been in use, the recesses should be

thoroughly degreased and cleaned before brazing. The tips likewise should be cleaned in hydrochloric acid and carbon tetrachloride. Where grit blasting equipment is available, it is excellent practice to grit blast

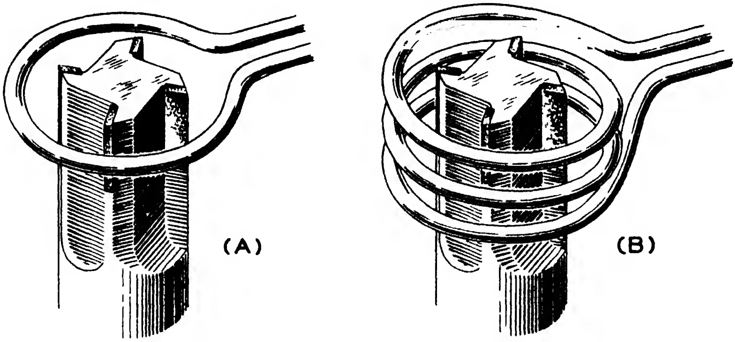


Fig. 12. Setup for Brazing in a Single-Turn Induction Coil (A) and in a Multiple-Turn Coil (B)

the surfaces to be brazed. It has been found that surfaces which have been blasted to a satin finish permit better capillary action of the molten brazing material placed between them.

Brazing of tips to such milling cutter bodies can be done with the oxyacetylene torch but this is a slow and tedious process because of the close spacing of the teeth and because of the different materials of which the body is made. Since different materials expand unevenly when heated, there is an excellent possibility of the body warping and cracking under such treatment. For these reasons, the induction method of heating is the better choice.

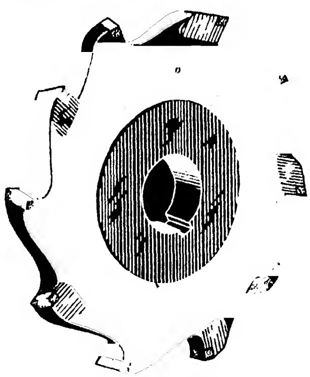


Fig. 13. A Milling Cutter Having a Body Made of Steel, Meehanite, or Other Cast Iron

When the spaces between the teeth are broad enough to permit the insertion of a single-turn coil between them, the brazing may be done by induction. The setup for brazing by this method is shown in (A) of Fig. 14, while the coil itself is shown in (B). This coil permits the setting or puddling of the tip into the recess where it can be held until the brazing material has had a chance to solidify.

In (C) of Fig. 14 is shown another type of induction coil which is adjustable to the width of the cutter. The two tubular rods holding the blocks form the single-turn induction coil. In all cases of induction heating, the coils are cooled by circulating water.

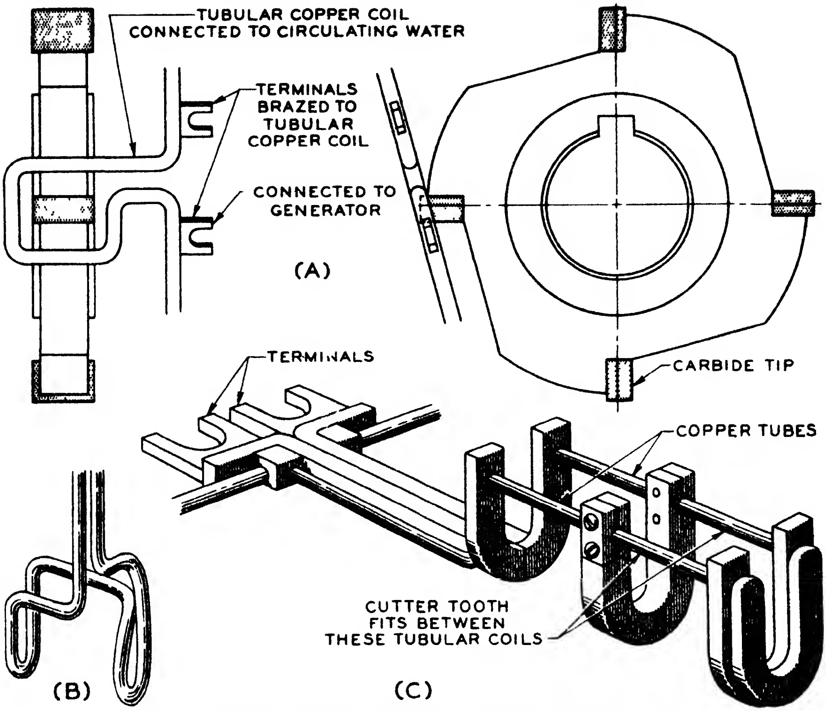


Fig. 14. Specialized Induction Brazing Coils for Milling Cutters

Brazing Combination Tools. Combination tools, such as the one shown in Fig. 15, are designed for reaming, countersinking, and spot-facing of a stepped hole. They can be brazed with the oxyacetylene torch in much the same way as the reamer or the milling cutter, each tip being brazed separately. They can also be brazed in the furnace when the tips are held in place with graphite rods and tied with nichrome wire.

Induction brazing of such tools also may be done successfully either with a single-turn coil or a multiple-turn coil especially designed for the job. The turns of the coil must conform to the diameter of the tool.

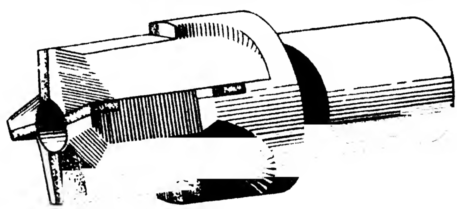


Fig. 15. A Combination Tool Which Presents a More Complicated Brazing Problem

Recapitulation. Whatever method of heating for brazing is employed, the essentials necessary for good, sound brazes are well-prepared surfaces, cleanliness of the surfaces, use of the proper flux and

brazing material, and careful heating. If these precautions are observed, satisfactory brazes will be obtained no matter what method is employed.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What is milling?
2. What is a milling cutter?
3. State the difference between the axial and the radial rake angle of the milling cutter.
4. What is the function of the primary relief on the milling cutter tooth?
5. For what reason is a chamfer ground on the tooth?
6. Why is the facing cutter tooth ground to a bevel angle?
7. What useful function, if any, is performed bythe concavity of the tooth on the facing mill?
8. What three methods are used for brazing carbide tips to milling cutter bodies?
9. In torch brazing, why is it desirable to cover with a copious amount of flux, the region of the cutter that is being heated.
10. Why are the cutter bodies and the tips degreased before brazing?
11. Would you use the torch method of brazing tips on a four-fluted reamer 5/8'' in diameter? Why?
12. Why is the chip produced in milling of varying thickness?
13. What is the land of a milling cutter tooth?
14. Does the land grow larger or smaller as the cutter is reground?
15. What is a stagger-tooth cutter?
16. Should an eight-tooth milling cutter 6'' in diameter, of Meehanite and steel construction, be brazed?
17. Give two reasons why.
18. Why are copper-tube induction coils cooled by water circulation?
19. What is the one big advantage of a single-turn induction coil for the brazing of carbides?
20. What is the principal advantage of the multiple-turn induction coil?

REFERENCES

Boston, O. W. Metal Processing. New York: John Wiley & Sons, Inc., 1941.

Curtis, Frank W. High-Frequency Induction Heating. New York: McGraw-Hill Book Company, Inc., 1944.

Dow Chemical Co. ''Machining Magnesium,'' Dowmetal Data Book.

Handy & Harmon. "Brazing Carbide Tool Tips with Easy-Flo #3," Bulletin 11-A.

Kearney & Trecker Co. "Metal Cutting Research Data."

Metal Cutting Tool Institute. "Milling with Carbide." 1947.

King, Lewis E. *Milling Machine Operations*. New York: The Macmillan Company, 1944.

National Twist Drill and Tool Company. *Milling Cutters and Milling*.

Setapen, A. M. "Silver Alloy Brazing with Induction Heating," *Transactions of the Electrochemical Society*, Volume 86, 1944.

CHAPTER XI

Sharpening Milling Cutters

Importance of Cutter Maintenance. Every carbide milling cutter must be properly ground when it is made, and must be kept properly ground throughout its useful life. Many of the principles outlined in Chapter V on the grinding of single-point carbide tools apply here. But since milling cutters turn with or against the feed of the work, certain added problems arise in their use. In this chapter, the latest data will be given on all these matters in as simple a form as possible. Old principles are applied to new jobs and recently discovered techniques are discussed which have developed from the tremendous progress made in recent years.

Experience shows that properly sharpened carbide-tipped milling cutters produce finer surfaces on the work and last much longer under actual conditions of use than do tools which are dull. The dull cutter also requires more power for its operation and exerts more of an impact force on the work, on the machine, and on the cutter arbor. This pounding effect is detrimental to the tool itself as well as to the machine.

Production efficiency of the carbide milling cutter, as well as any other milling cutter, is largely dependent upon the keenness of its cutting edges. This keenness can be maintained by proper and timely sharpening. A dull cutter, just as a dull lathe tool, wears more rapidly than a sharp one, producing poorly finished surfaces and consuming more power in doing the work. When a tool which has been allowed to become excessively dull is sharpened, it will be found necessary to remove more of the carbide stock in restoring it to the required keenness than would have been necessary had the tool been resharpened more frequently. While the frequency of cutter sharpening is a matter which is difficult to establish by hard and fast rules, good practice dictates that at the first sign of cutter wear, indicated by a poorly finished surface or greater power requirements, or both, the cutter should be sharpened. The frequently sharpened cutter will outlast those not sharpened until their cutting edges are badly worn. For this reason, it is advisable to grind the cutters when the wear on the flank of the tooth extends to not more than 1/32". This rule is considered by many tool engineers to be the best guide that has been established.

Classification of Cutters. To avoid confusion as to the designations of carbide-tipped milling cutters, a classification and brief

description is given of those tools in general use. A list of these types follows:

- plain milling cutters
- stagger-tooth milling cutters
- side milling cutters
- face mills
- shell-type face milling cutters

A plain milling cutter in a gang setup is shown in Fig. 1. All of the cutters pictured in this illustration could have carbide tips brazed to the

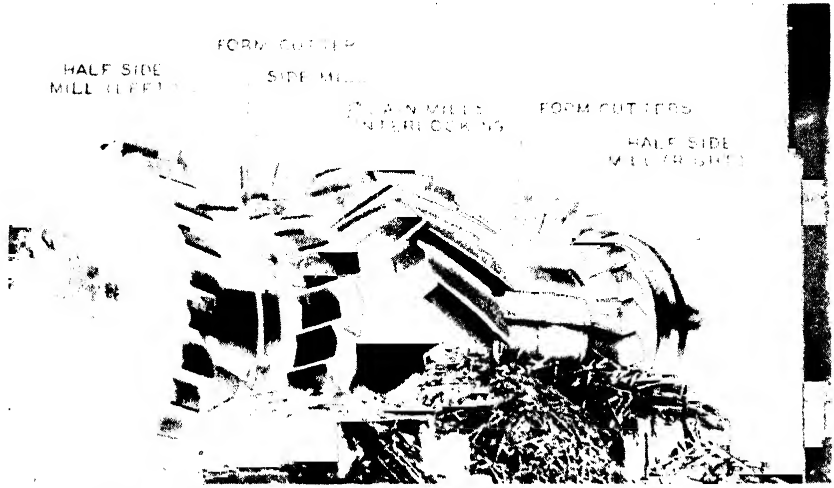


Fig. 1. A Setup for Gang Milling, Using Three Side Mills; Right and Left Interlocking Spiral-Type Cutters, Giving a Stagger-Tooth Effect; and Two Forming Cutters
Courtesy of Brown & Sharpe Mfg Co

steel bodies. The plain type of cutter is used for milling straight surfaces, keyways, and grooves. The stagger-tooth milling cutter, described and illustrated in Chapter X, is used extensively for milling slots. Its advantage is that the sides of a groove cut are finished by every other tooth, thus eliminating some of the side rubbing action.

Fig. 2 shows a right and left setup of milling cutters, a combination frequently used for facing work on two sides at one time. Actually, these cutters are similar in action to face milling cutters. They are simply mounted in pairs on the arbor of the machine. Those shown have inserted teeth which are held in place by a special locking device. Illustrations in latter portions of this chapter will show face mills with carbide tips which have been brazed to the bodies of the cutters. The inserted tooth type of cutter may have either solid carbide inserts wedged or clamped on the cutter body, or the blades may be made of a suitable steel and tipped with carbide, then wedged or clamped to the

tool. Insert-type face mills with solid carbide teeth are preferred by many tool engineers.

Shell-type face mills are mills of smaller size. These are usually mounted on a taper shank or on the spindle of the vertical milling ma-

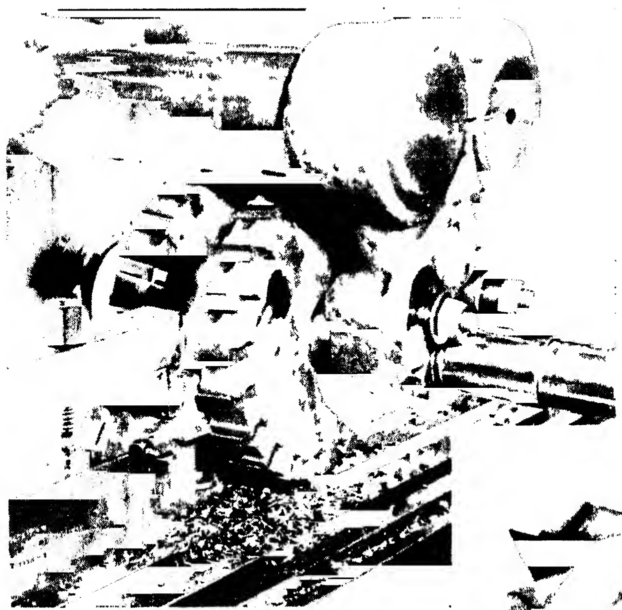


Fig. 3 *(Below)* Shell-Type End Mill
Courtesy of Vascoloy Ramet Corp

Fig. 2. *(Above)* Left and Right Setup of Milling Cutters for Facing Two Sides of the Work in One Operation
Courtesy of the McCrosky Tool Corp

chine attachment. They usually have sintered or cast alloy blades which have been brazed to the teeth. The construction of a common shell-type end mill is shown in Fig. 3. The "fly" type with one, two, or three teeth was illustrated in Chapter II.

Relief Angles of Teeth. The primary relief of the milling cutter is ground behind the cutting edge. The grinding may be done either with a straight wheel or a cup wheel as shown in (A) and (B) of Fig. 4. A carbide-tipped milling cutter ground by either of these methods will be found satisfactory and can be made to produce any desired finish on the part being machined, provided certain basic principles of grinding are observed.

Correct relief of the cutting edge is essential, because teeth with in-

sufficient relief will drag over the work, causing frictional heat and poor finish. At the same time, additional power will be required to cut the metal and a more rapid dulling of the cutting edges will be noted. On the other hand, too much relief will produce chatter and rapid tool wear. The relief angles on carbide-tipped milling cutters are not standardized. However, certain angles are recommended for specific materials and operating conditions. Those generally recommended have been determined through practical experiments and are given in Table I.

The angle or relief on the cutter usually can be measured with a protractor. However, when the land behind the cutting edge is wide, a protractor measurement will not tell the whole story, for it is then

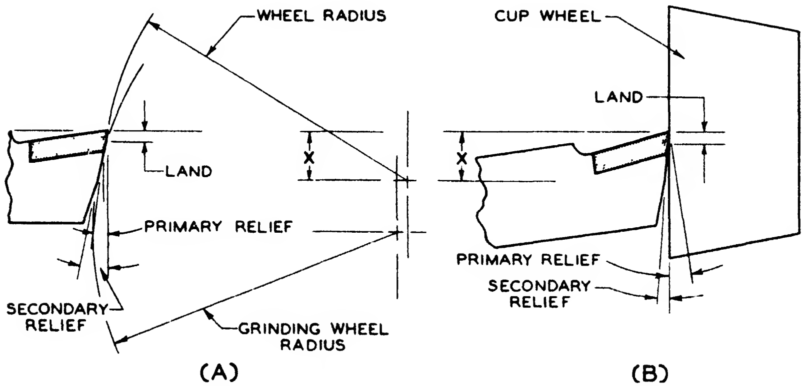


Fig. 4. Satisfactory Grinding of Carbide-Tipped Cutters May Be Done Either with the Straight Wheel (A) or the Cup (B)

possible to have the correct relief angle, yet have the back of the tooth rub the work. This condition is illustrated in Fig. 5. If the back part of the tooth is to clear the work, that portion of the tooth must be ground off enough to provide a secondary clearance. This clearance also is given in Table I.

Another method of measuring the desired cutting relief on the periphery of the cutter is to measure the drop of the line behind the cutting edge, 1/16" from the edge, with a dial indicator. This procedure is shown in Fig. 6. This method of measurement is more accurate than the protractor reading and is preferred by most tool engineers and inspectors. Another advantage of this method is that the secondary relief can be checked at the same time merely by rotating the cutter through the greater angle. The values of the reading drop in thousandths of an inch on the indicator dial are also shown in Table I.

The desired drop of the indicator pointer can be obtained in grinding by setting the index finger, or the so-called tooth rest, below or above the center of the grinding wheel or the cutter by the amounts specified in Tables III and IV. The methods of setting the tooth rest will be

TABLE I. RECOMMENDED RELIEF AT CUTTING EDGE OF CARBIDE-TIPPED MILLING CUTTERS

MATERIAL TO BE MACHINED	RELIEF AT CUTTING EDGE		
	Primary Relief in Degrees	Secondary Relief in Degrees	Indicator Drop in Inches Per 1/16 Inch
Aluminum	8 to 9	10 to 15	.009 to .010
Brass and bronze			
hard	5 to 6	8 to 10	.005 to .006
soft	5 to 6	8 to 12	.005 to .006
Cast iron			
hard	5 to 6	8 to 10	.005 to .006
soft	5 to 7	8 to 12	.005 to .007
Semisteel	5 to 6	8 to 10	.005 to .006
Copper	8 to 9	12 to 14	.009 to .010
Fiber	8 to 9	12 to 14	.009 to .010
Malleable iron			
hard	5 to 6	8 to 10	.005 to .006
soft	5 to 7	8 to 12	.005 to .007
Magnesium	8 to 9	10 to 15	.009 to .010
Plastics	8 to 9	10 to 14	.009 to .010
Rubber			
hard	8 to 9	12 to 14	.009 to .010
medium	10 to 12	14 to 16	.011 to .013
Steel			
rolled carbon	6 to 8	8 to 12	.006 to .007
rolled alloy	5 to 7	8 to 12	.005 to .007
cast	6 to 7	9 to 12	.006 to .007
Zinc alloys, die cast	9 to 10	12 to 15	.010 to .011

explained in a more detailed manner in subsequent pages of this chapter.

Grinding the Relief. The grinding of the relief angles of a milling cutter was illustrated in (A) and (B) of Fig. 4. This grinding can be done with a straight wheel or a cup wheel. When the land behind the cutting edge is small; plain grinding wheels produce very satisfactory cutting edge conditions. When the lands become larger, cup wheels are used. Unless the wheel was very large or was set at an angle, a straight wheel would grind the teeth too concave.

When grinding with a straight wheel, the relief behind the cutting edge may also be too excessive. An exaggerated drawing of this condition is presented in Fig. 7. This could not happen if the relief were ground with a cup wheel.

Width of Land on Teeth. Width of the land on the cutter teeth

varies with the size of the milling cutter and the material to be cut. The width for plain milling cutters is from 1/32'' to 1/16''. It is on this land that the primary relief is ground, and the width of the land increases

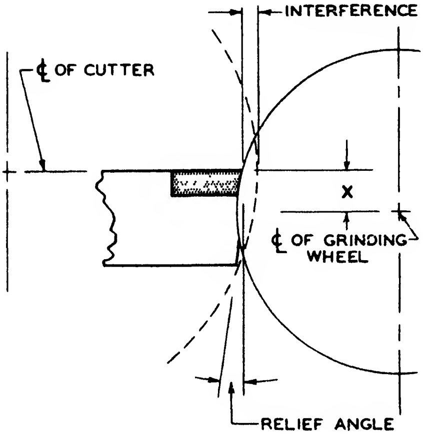


Fig. 5. Interference of the Back Part of the Tooth with the Work

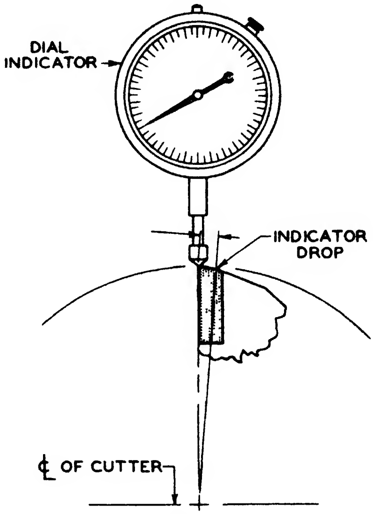


Fig. 6. Use of the Dial Indicator in Measuring the Relief

with each grinding. When the land width becomes too great, it must be reduced by grinding the secondary clearance angle. For smaller tools such as end-cutting face mills and end mills, the best performance is obtained when the lands are as indicated in Table II.

Grinding Plain Milling Cutters. To sharpen the teeth of a plain milling cutter tipped with carbide, the cutter is mounted on a mandrel and set between centers in the grinding machine, or the cutter is mounted on a special stub arbor and is held in a universal swivel fixture, as shown in (A) of Fig. 8. After the cutter has been properly mounted in the grinding machine, the tooth rest, which can be seen in both (A) and (B) of Fig. 8, is mounted either on the swivel base of the work holding fixture, as in (A), or on the work-holding head or fixture, as in (B). It is adjusted to that tooth which is to be ground first. The table of the machine and the tooth rest are adjusted so that the grinding of the land behind the cutting edge will result in the proper relief for each tooth.

The cutter is then brought carefully in contact with the rotating grinding wheel. The table of the grinding machine is moved forth and back until the grinding wheel has finished one tooth. At this point, the tooth is checked for taper. This is done by holding the work between centers, and checking with a dial indicator as illustrated in Fig. 9. When

a dial indicator is not available, it is customary to rotate the cutter in the grinding machine 180° and, without changing the depth of the cut of the grinding wheel, make a trial cut on the tooth which is just opposite the one previously ground. The measurement can then be made with a micrometer or a vernier caliper.

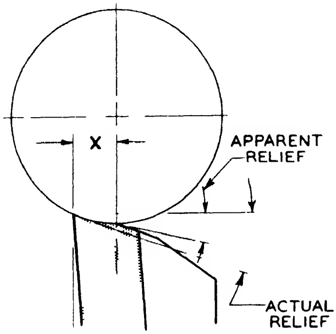


Fig. 7. The Relief on a Cutter When Ground with a Straight Wheel

Should taper be found on the cutter, the table of the machine must be readjusted so that the cutter, after grinding is perfectly cylindrical. If there is no taper, the teeth of the cutter are finished, one by one. The cutter is revolved backwards for indexing, the tool rest engaging each successive tooth to be sharpened. This procedure is repeated until all teeth are finished.

During the process of sharpening, the grinding wheel will wear. Therefore, to compensate for this wear, finishing cuts should be started 180° from the tooth at which the first roughing cuts were made. For final finishing, a cut removing not more than .0005'' should be made on each tooth.

TABLE II. LANDS ON END MILLS AND SMALL FACE MILLS TIPPED WITH SINTERED CARBIDE

Material Milled	Width of Lands for Cutters in Inches							
	Roughing Cuts				Finishing Cuts			
	3/8-1/2	9/16-1	1 1/16-2	2 1/16-4	3/8-1/2	9/16-1	1 1/16-2	2 1/16-4
Aluminum and magnesium	.010	.020	.030	.040	.07	.012	.015	.020
Brass and bronze	.032	.045	.062	.062	.020	.030	.045	.062
Cast iron								
soft and medium	.032	.045	.062	.062	.020	.030	.045	.062
hard and chilled	.045	.045	.062	.093	.030	.045	.062	.093
Copper	.010	.015	.020	.030	.007	.010	.015	.020
Fiber	.010	.015	.020	.030	.007	.010	.015	.020
Malleable castings	.030	.045	.062	.062	.020	.030	.045	.045
Plastics	.007	.010	.015	.020	.005	.007	.010	.015
Rubber								
hard	.010	.015	.020	.030	.007	.010	.015	.020
soft	.005	.007	.012	.015	.003	.005	.008	.010
Steel								
soft	.020	.025	.030	.045	.015	.020	.030	.040
hard	.030	.045	.062	.093	.020	.030	.045	.062
White metals, die cast	.010	.015	.020	.030	.007	.010	.020	.030

Direction of Wheel Rotation. In grinding carbide-tipped milling cutters, the direction of grinding wheel rotation, in relation to the cutting edge of the tool, is very important. There are two methods of

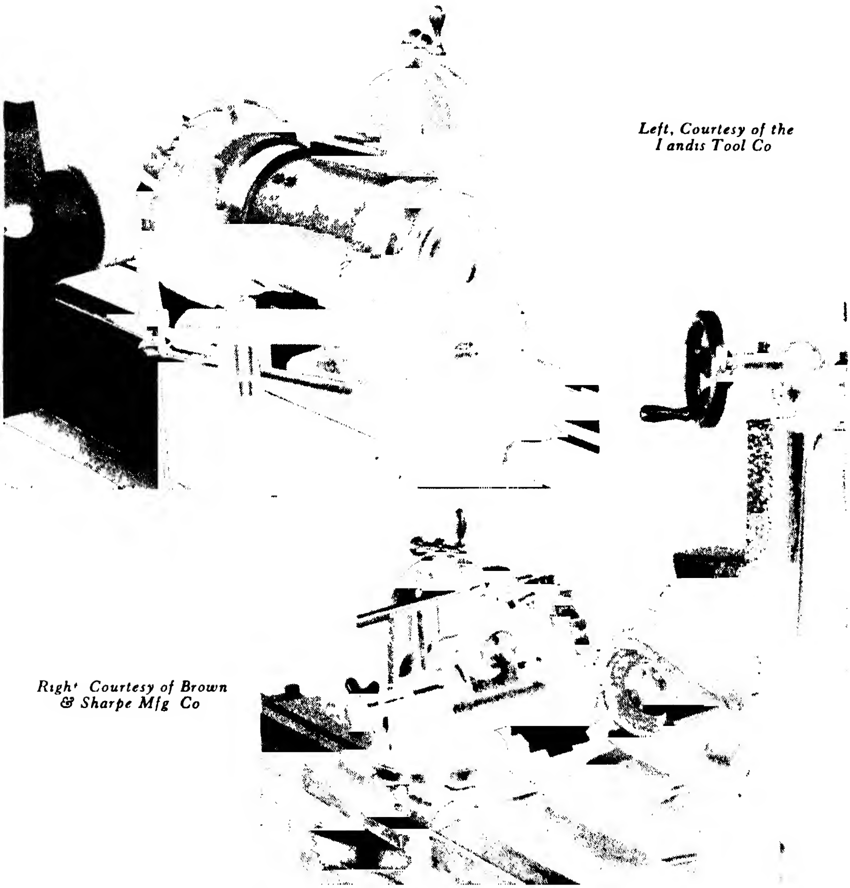


Fig. 8. The Setup for Grinding Plain Milling Cutters Tipped with Carbide (A) *(Upper Left)* and Details of the Setup in a Universal Swivel Fixture (B) *(Lower Right)*

grinding cutters in general cutter sharpening practice, regardless of whether grinding is done with a cup or a straight wheel. The two methods are based on the direction of rotation of the grinding wheel in relation to the cutting edge of the tooth. These methods are illustrated in (A) through (D), inclusive, of Fig. 10.

In the method shown in (B), the pressure of the grinding wheel is against the tooth rest and from the body of the carbide toward the cutting

edge. This method of grinding a carbide-tipped tooth is not recommended, for it has a tendency to pull out particles of carbide from the cutting edge, leaving it rough. Its chief advantage, however, lies in the stability of the cutter during grinding operation. When the grinding pressure is against the tooth rest, there can be no possibility of the tool turning during grinding, thus spoiling the tooth being ground.

In the method shown in (A), the grinding wheel rotates so that cutting is from the cutting edge toward the body of the tooth. Here, the pressure of the grinding wheel tends to rotate the cutter away from the tooth rest. It should be noted that unless the operator exercises great care in holding the cutter against the rest, it is possible for the tool to rotate, producing a ruined tooth.

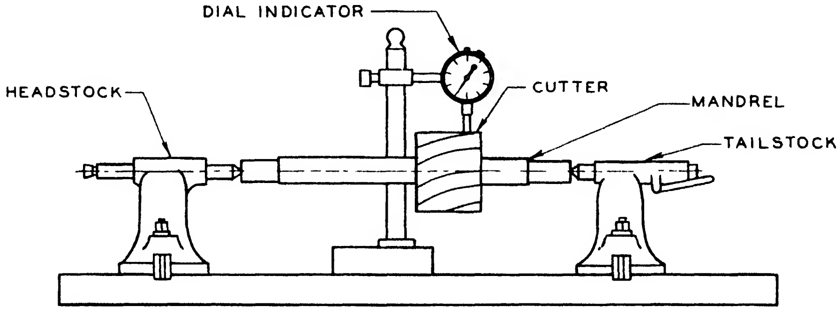


Fig. 9. Checking the Tooth for Taper

The same setups may be used when grinding is done with the straight wheels as illustrated in (C) and (D) of Fig. 10. However, as pointed out before, all carbide tools should be ground with the cut being made from the edge toward the body of the tool. This is done in order to avoid chipping the edge of the hard carbide tip. This can only be done when the cutter is ground as shown in (A) and (D) of Fig. 10. If the cut is light and the operator has the necessary experience, the danger of spoiling the cutter is slight.

When a considerable amount of material is to be removed from the teeth of a milling cutter, rough grinding may be done by the setups shown in (B) and (C) of Fig. 10 in which the wheel pressure is against the tooth rest. This allows adequate support for the increased pressures resulting from the added quantity of material being removed. Following the roughing cuts, the cutter should be reset and reground for finishing by the methods shown in (A) and (D). That is, grinding should be done from the cutting edge of the cutter toward the back. This method of grinding will produce keen cutting edges.

Selection of Grinding Wheels. For sharpening carbide-tipped milling cutters, grinding wheels must be selected with care. Satisfactory grinding results depend largely on the care with which the choice is made. A soft or medium-soft grade silicon carbide, free-cutting

wheel is preferred over the harder grades. The softer wheel cuts more freely and is not so likely to glaze and overheat the cutting surfaces and edges of the carbide-tipped teeth.

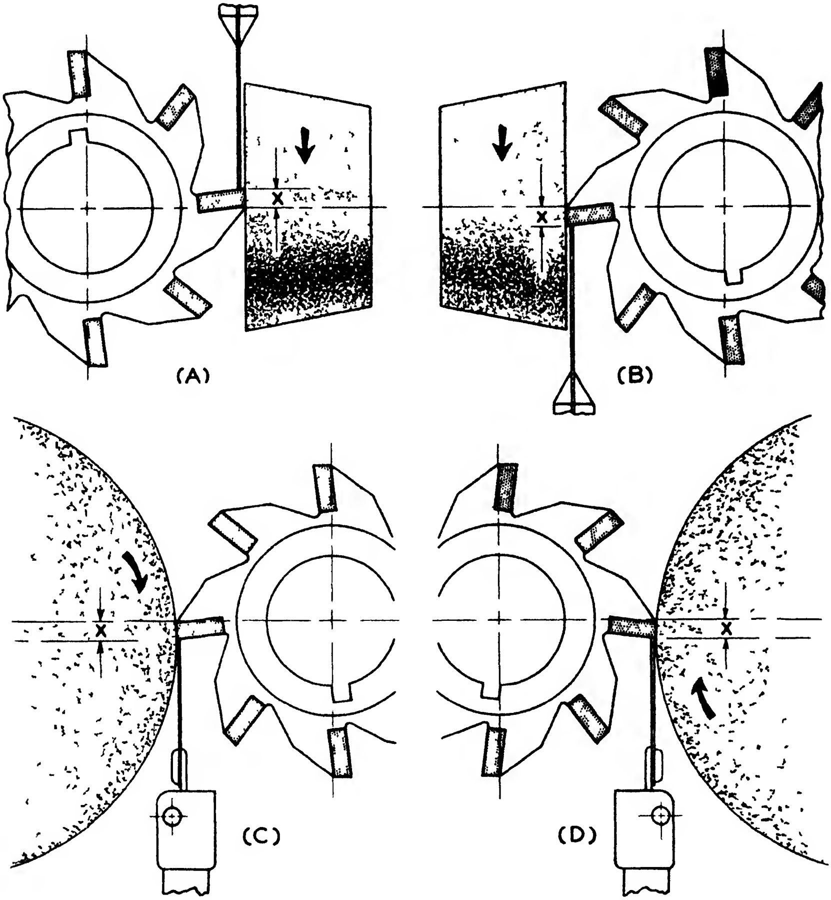


Fig. 10. The Direction of Grinding Wheel Rotation in Relation to the Cutting Edge of the Tool Is Extremely Important

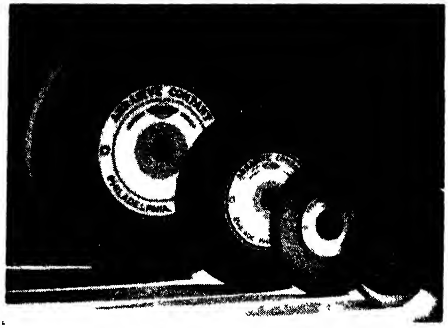
Wheels used for sharpening carbide-tipped cutters are the flared cup and the straight types. Typical wheels are shown in (A) and (B) of Fig. 11. The flared cup type of wheel is preferred by many toolroom men for all sharpening of carbide-tipped milling cutters.

When only a relatively few tools are to be sharpened, roughing cuts on milling cutters are best made with a vitrified bond, silicon-carbide wheel of a grain size from 80 to 100, F or G grade of medium density.

These wheels are capable of removing from .001" to .002" per pass. The finishing cut, of course, should be taken with a much finer wheel. For long tool life, better performance, and better finish of the work surface, carbide-tipped milling cutters should be finish ground with a 180 grit silicon-carbide wheel, or, better yet, with a 180 to 220 grit diamond wheel of 100 concentration. The latter gives a very keen cutting edge to the carbide cutter.

Right Courtesy of the Abrasive Company

Below, Courtesy of the Norton Company



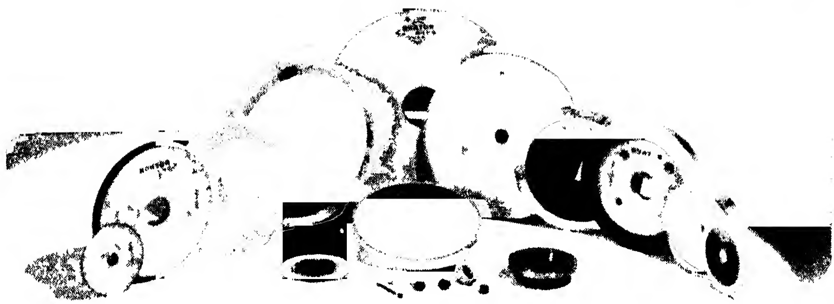


Fig. 11. Typical Grinding Wheels. In (A) *(Upper Right)* Are Shown Silicon Carbide Wheels. In (B) *(Below)* Are Shown the Diamond Wheels.

Where the number of cutters to be ground is large enough to warrant investment in diamond grinding wheels, the roughing should be done with 100 grit, resinoid-bond diamond wheels, and finishing either with a 220 or 400 grit diamond wheel, preferably of 100 diamond concentration. Diamond wheels cut cooler than the silicon-carbide wheels. There is less danger, therefore, of cracking the carbide tips when diamond wheels are used for sharpening.

It should be remembered that a fine grain wheel should not remove more than .0002" to .0003" per pass. Attempts to remove more material per pass may result in overheating and cracking of the cutting edges of the teeth.

Setting Rest for Relief Angles. The relief angle, or the clearance immediately behind the cutting edge, is the product of the proper location of the grinding wheel, the cutter, and the tooth rest, each with relation to the other. Tooth-rest blades in common use are shown in (A), (B), and (C) of Fig. 12. A complete tooth-rest assembly is shown in (A) of Fig. 13. There are different methods of making such set-ups, depending on the type of the wheel used, the shape of the cutter, and the location of the tooth rest. The tooth rest may be located on the table, on the fixture holding the cutter, or on the head of the grinding machine.

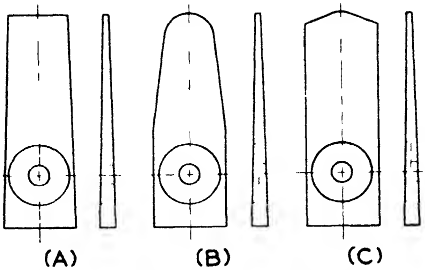


Fig. 12. Three Types of Tooth-Rest Blades Commonly Used in Cutter Sharpening

When setting up the work, the center of the cutter and the grinding wheel are brought to the same plane by adjustment of the table or the wheel head, or both. This adjustment is made with the aid of a center gage as shown in (A) and (B) of Fig. 13. The tooth rest is then set to give the desired amount of relief.

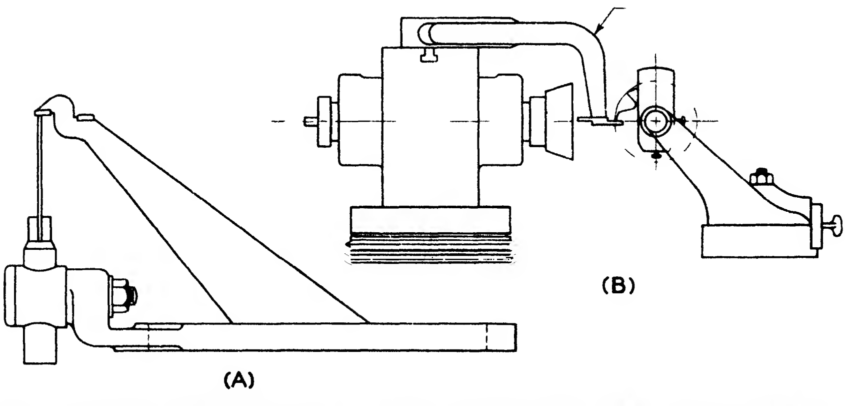


Fig. 13. Tooth Rest Assembly (A), and a Centering Gage for Setting the Cutter in the Center of the Wheel Spindle (B)

When using the cup-shaped wheel, the wheel center, the tooth rest (which may be mounted on the wheel head, as shown in Fig. 14), and the cutter center are brought to the same plane with the aid of the gage shown in (B) of Fig. 13. If the tooth rest is located on the wheel head as shown in Fig. 14, the wheel head may then be lowered or the table raised to give the cutter teeth the desired relief.

The distance the wheel head with the tooth rest is to be moved from the center position will depend upon the size of the cutter and the de-

sired clearance angle. This distance, designated by the letter X, was shown in Figs. 4, 5, and 10.

When the straight wheel is used for cutter sharpening, the center of the cutter, the center of the wheel, and the tip of the tooth rest mounted on the table, are brought into the same horizontal plane by the use of a

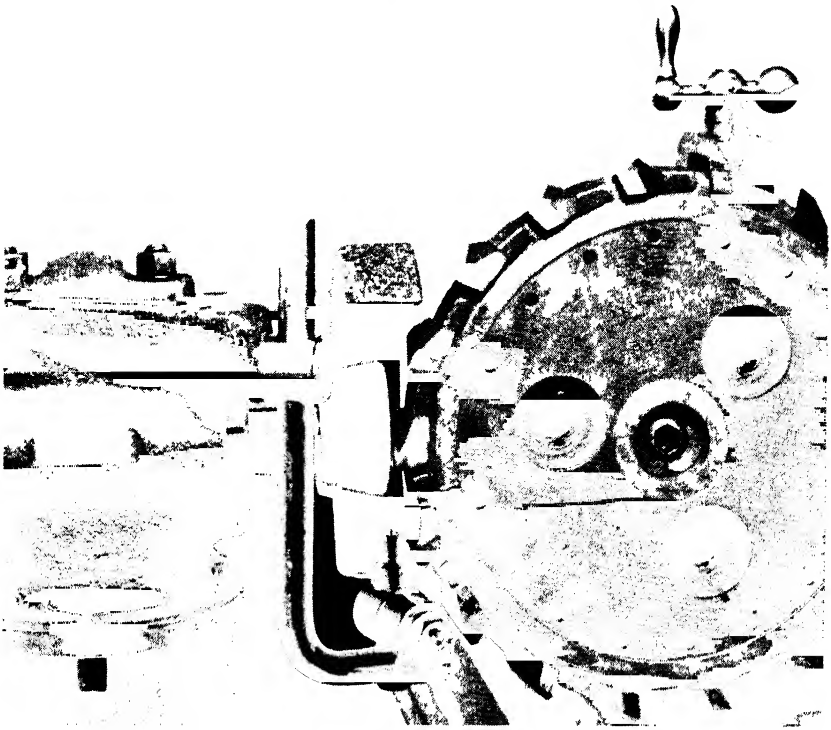


Fig. 14. Cup Wheel Setup for Grinding Carbides
Courtesy of R K LeBlond Machine Tool Co

center gage. The table with the tooth rest is then lowered to give the desired relief to the cutter. The distance the table will have to be lowered will depend on the relief angle desired and upon the diameter of the grinding wheel and not the cutter as was the case before. This is shown in (C) and (D) of Fig. 10. Same results are obtained by lowering the tooth rest distance X shown in (C) and (D) of Fig. 10.

Rules for Setting the Cutter. To determine the setting of the cutter when a flared cup wheel is used for grinding, multiply the diameter of the cutter by the relief angle and by .0087, or multiply the radius of the cutter by the number of degrees desired for the relief, and the tangent of 1°. Stated as a formula, it becomes

$$X = D \times A \times .0087$$

in which

X = the setting of the cutter above the center in inches

D = the diameter of the cutter in inches

A = the desired relief angle in degrees

.0087 = a constant

TABLE III. SETTING TOOTH REST FOR GRINDING WITH CUP WHEELS

Cutter Diam. in Inches	Values of X in Inches for Relief Angles Shown								
	3°	4°	5°	6°	7°	8°	9°	10°	12°
1/2	.013	.018	.022	.026	.030	.035	.039	.043	.052
3/4	.020	.026	.033	.039	.046	.052	.059	.066	.078
1	.026	.035	.044	.052	.061	.070	.078	.087	.104
1 1/2	.039	.052	.066	.078	.092	.104	.117	.131	.156
2	.052	.070	.088	.105	.122	.140	.157	.175	.208
2 1/2	.065	.087	.110	.137	.152	.175	.197	.218	.260
3	.078	.105	.131	.157	.183	.210	.235	.261	.312
3 1/2	.092	.152	.152	.183	.214	.244	.274	.305	.364
4	.105	.140	.175	.210	.244	.279	.314	.349	.416
4 1/2	.118	.157	.196	.236	.275	.314	.353	.392	.464
5	.131	.175	.220	.262	.306	.349	.392	.435	.520
5 1/2	.144	.192	.240	.288	.336	.384	.432	.480	.572
6	.151	.210	.262	.316	.366	.420	.471	.522	.624
7	.182	.244	.305	.366	.427	.487	.5481	.608	.728
8	.209	.279	.349	.418	.487	.557	.626	.695	.832

To check the operation of this formula, assume it is desired to determine the setting of the tooth rest below center as shown in (B) of Fig. 10, or of the tooth rest above the center of the wheel and the cutter as shown in (A) of Fig. 10, for a 6'' milling cutter with a 5° primary relief. Grinding is to be done with a flared cup wheel. The diameter, D, is 6'', and the relief angle, A, is 5. Substituting these values in the formula results in

$$X = 6 \times 5 \times .0087 = .261$$

When the tooth rest is mounted on the table of the grinding machine, the tooth rest is lowered the required amount for relief, .261 in this problem. This procedure was shown in (B) of Fig. 10.

When the tooth rest is mounted on the grinding wheel head, the wheel head may be raised or the table with the cutter lowered. This is shown in (A) of Fig. 10.

To determine the setting of the cutter when the plain or straight wheel is used, multiply the diameter of the wheel by the relief angle and by .0087 as before. The result is the amount of the setting above or below center. The formula just given can be used by substituting the diameter of the wheel for the diameter of the cutter. For example, if it were desired to determine the setting of the cutter for 5° of primary relief, using an 8'' grinding wheel, when A was 5, D was 8, and the constant .0087, the formula with the substitutions made would be

$$X = 8 \times 5 \times .0087 = 0.348''$$

When the tooth rest is mounted on the table, the wheel head is raised or the table with the work is lowered the required amount. This is indicated by X in (C) and (D) of Fig. 10. Thus, the tooth rest is kept in the center line of the cutter.

In general, the tooth rest is fastened to the table when grinding cutters with straight teeth. For stagger-tooth cutters with angular teeth, the tooth rest should be mounted on the grinder head if the work is placed on a mandrel and held between centers.

Values for setting the tooth rest for cup wheels for 3° to 10° relief angles on cutters from 1/2'' to 6'' in diameter are shown in Table III. Values for setting the tooth rest for straight grinding wheels for 3° to 10° relief and for diameters of wheels from 2'' to 8'' are given in Table IV.

Location of the Tooth Rest. In (A), (B), (C), and (D) of Fig. 10 is illustrated the location of the tooth rest in relation to the rotation of the grinding wheel. The tooth rest may be set from above or below, depending on whether it is attached to the work head or to the table of the machine. Actual locations of some tooth rests on the machine were shown in (A) and (B) of Fig. 8. In both cases, however, the tooth rest is fastened to the work head which, in turn, is mounted on the table.

Fig. 14 shows a setup with the tooth rest mounted on the grinding head. This type of mounting is used when the cutter has helical teeth. That is, teeth which are set at an angle to the axis of rotation. Such a mounting lets the tooth slide on the tooth rest while the grinding is going on, thus assuring a uniform relief over the entire width of the tooth. For grinding on the outside diameter of a cutter that has inclined or helical teeth, a tooth-rest blade similar to that shown in (C) of Fig. 12 is used. Such tooth-rest blades should conform to the angularity of the teeth to be ground. The same type of tooth-rest blade is used for sharpening stagger-tooth milling cutters in which alternate

TABLE IV. SETTING TOOTH REST FOR GRINDING WITH PLAIN WHEELS

Diam. of Grinding Wheel in Inches	Values of X in Inches for Relief Angles Shown								
	3°	4°	5°	6°	7°	8°	9°	10°	12°
2	.052	.070	.088	.105	.122	.140	.157	.175	.208
2 1/2	.065	.087	.110	.131	.153	.175	.197	.218	.261
3	.078	.105	.131	.157	.183	.210	.235	.261	.312
3 1/2	.092	.122	.152	.183	.214	.244	.274	.305	.364
4	.105	.140	.176	.210	.244	.279	.314	.349	.416
4 1/2	.118	.157	.196	.236	.275	.314	.353	.392	.468
5	.131	.175	.220	.262	.306	.349	.392	.435	.520
5 1/2	.144	.192	.240	.287	.336	.384	.432	.480	.572
6	.157	.210	.262	.316	.366	.420	.471	.522	.624
6 1/2	.172	.228	.286	.344	.400	.453	.510	.567	.675
7	.185	.246	.308	.370	.430	.488	.550	.610	.730
7 1/2	.198	.264	.330	.396	.462	.523	.588	.654	.780
8	.211	.282	.352	.422	.492	.558	.627	.698	.835

teeth are inclined in the opposite direction or have right and left helix angles.

As has been pointed out before, grinding of carbide-tipped milling cutter teeth can be done with wheels cutting toward the cutting edge or away from it, toward the body of the tip. The first method should be used for roughing cuts only. The tool should be finish ground by the second method for best results. A good rule to remember in grinding all carbide tools, including milling cutters, is to grind so that the cut is from the edge toward the body. Only then will a satisfactory tool be obtained with a minimum chance of damaging the cutter.

The tooth rest, regardless of the direction of rotation of the grinding wheel, is always set on the face of the tooth which is being ground, and as close to the cutting edge as is possible. This is done so as to assure the tooth as rigid a support as is possible. However, this support is never very rigid, for the tooth-rest blades are usually made from materials 3/64" to 1/16" thick and 1/2" wide, a construction which is bound to be somewhat springy. Locating the tooth rest on the back of the tooth or from the face of another tooth is not recommended because the spacing between teeth is seldom accurate, and grinding by locating from adjacent tooth would result in a poorly finished cutter.

Sequence of Grinding Steps. As a means of reviewing the foregoing material and making certain that a clear picture has been presented of the steps involved in sharpening a milling cutter, an out-

line of the procedure will be given of the actual grinding of a plain-tooth milling cutter. The problem is to set up and sharpen a 4" plain milling cutter which is to be used in machining aluminum.

First, it should be explained that for machining aluminum or magnesium, the cutter teeth should be ground very smooth, preferably with a 400 grit diamond wheel. The primary relief, as shown in Table I, should be 8° to 9° for best performances. When sharpening is done on a flared cup wheel either of diamond or silicon carbide, the value of X is taken from Table III. This would be 0.279" for an 8° relief angle. The sequence of operations is as follows:

1. Select a proper tooth rest. Place it on the table and adjust it to the proper height with the aid of a height gage as shown in (A) of Fig. 13.
2. Mount the cutter on a mandrel and place it between centers on the grinding machine.
3. If the setup is similar to the one shown in (A) of Fig. 8, lower the tooth rest a distance of 0.279" (nearly 9/32") to give the cutter tooth the necessary clearance. The tooth rest should now be moved as close to the cutting edge as possible, keeping it just far enough away from the edge so as to prevent the grinding wheel from touching it.
4. Mount a 4" or 5" silicon carbide wheel, 80 grit, F or G grade, on the spindle of the machine and true the wheel with a diamond wheel dresser. A 100 grit diamond wheel of 100 concentration also could be used. However, it must be adjusted with a dial indicator so that it will run true. Use of the dial indicator in this respect was described and illustrated in Chapter V.
5. Set the machine so that the cutting speed will be between 4,000 and 5,000 f.p.m. Take a light cut on the tooth, moving the table with the cutter past the grinding wheel at a rapid rate (use the fast movement of the table actuating lever) and at the same time holding the edge of the cutter against the tooth rest.
6. Test the tooth thus ground for taper on angularity by the method shown in Fig. 6. If no taper is found, proceed to grind each tooth, removing from 0.001" to 0.002" per pass. Always take into consideration the fact that the silicon carbide wheel will wear rapidly in grinding tungsten carbide.
7. Proceed as described until all teeth have been ground. Then make certain that all teeth are equidistant from the center by taking a light cut on all of them with one setting of the wheel.
8. Examine the teeth through a magnifying glass. If all have been ground so that they are sharp and show no wear marks, change to a 180 grit diamond wheel and grind all teeth as before.
9. When all teeth have been rough finish ground, change to a 400 grit diamond wheel and finish grinding as before, holding the cutter tight against the tooth rest the while. In mounting the diamond wheel, make certain it runs true.
10. Using a dial indicator capable of measuring to .0001", check the cutter for runout. It should not be more than .0005". The method for

TABLE V. PERMISSIBLE RUNOUT OF MILLING CUTTERS

Diameter of Cutter in Inches	Cutter for Roughing Cuts		Cutter for Finishing Cuts	
	Face of Cutter	Outside of Cutter	Face of Cutter	Outside of Cutter
Up to 4	.001	.001	.0005	.001
4 1/8 to 10	.001	.002	.0005	.0015
10 1/2 to 15	.0015	.003	.001	.002
15 and over	.002	.004	.001	.003

testing the trueness of the wheel is shown graphically further along in the chapter. Permissible runout is shown in Table V.

11. Remove all loose metal particles from the cutting edge with a 400 grit diamond hone like the one shown in Chapter V. However, be certain that no part of the cutting edge is beveled or rounded out too much.

12. Inspect the surface finish of the cutter with a magnifying glass. If there are no imperfections, the cutter is ready for use.

Grinding Faces of Teeth. In the cutting of materials, the faces of the cutter teeth become cratered. That is, the chip digs out small pieces of carbide from the face of the tooth, leaving the face rough.

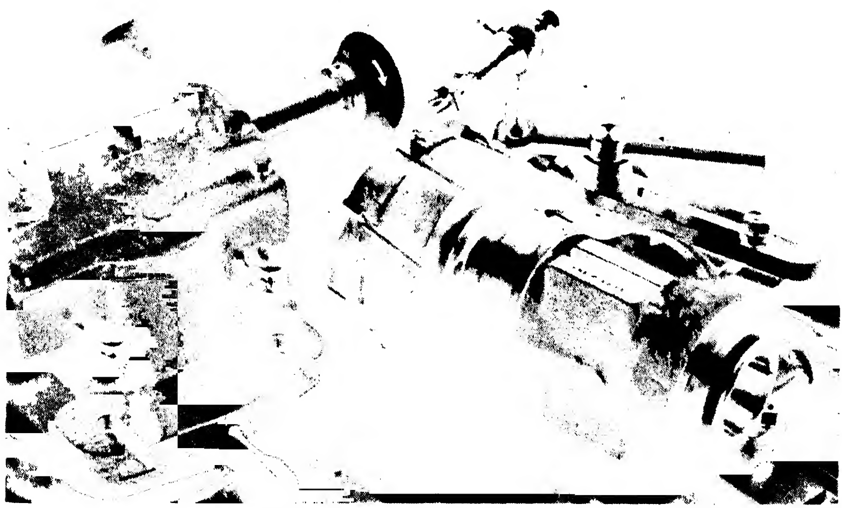


Fig. 15. Setup for Face Grinding of Cutter Teeth
Courtesy of Cincinnati Milling and Grinding Machines, Inc

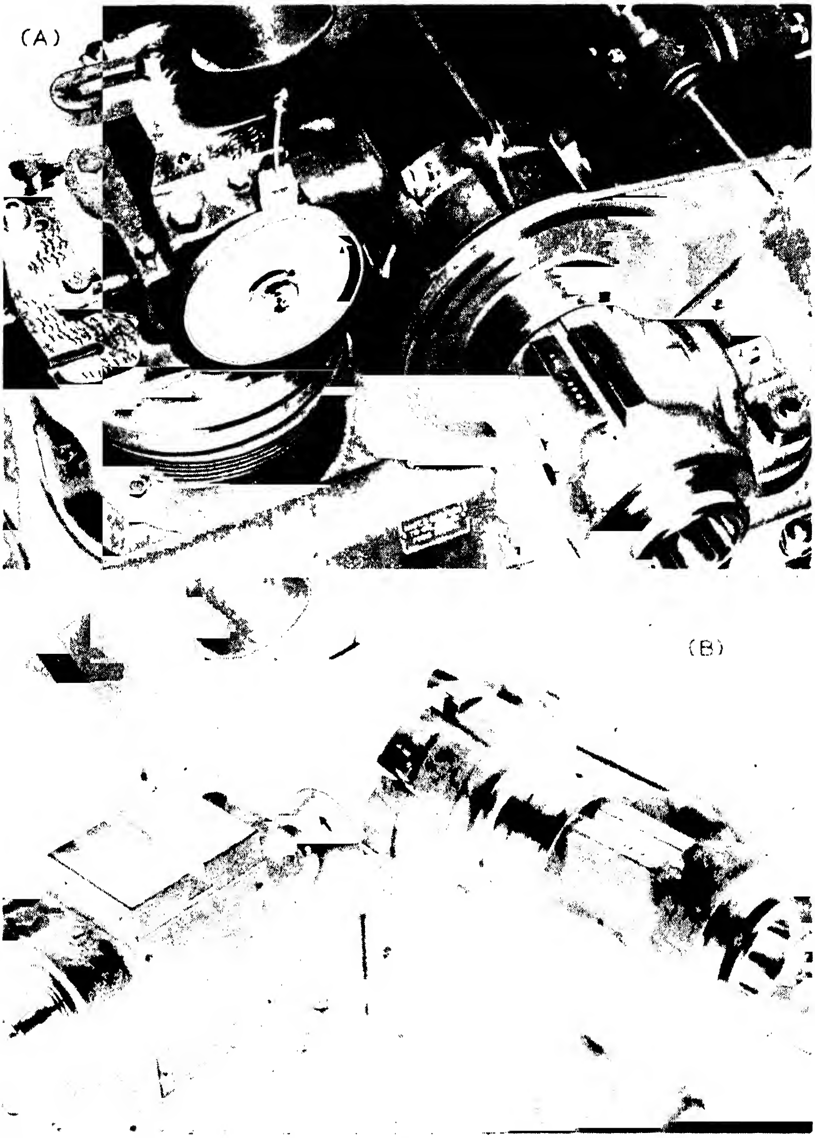


Fig. 16. (A) Grinding the Periphery of the Cutter After New Teeth Have Been Brazed In. (B) Grinding the Primary Relief

Courtesy of Cincinnati Milling and Grinding Machines, Inc

This happens when the cutter is machining tough, stringy material such as alloy steels. When this occurs, the faces of the teeth must be ground before the primary relief is established. This grinding is done by roughing with a 100 grit diamond wheel and finishing with a 180 to 220 grit resinoid-bond diamond wheel.

The setup for grinding the faces of the teeth on a face mill is shown

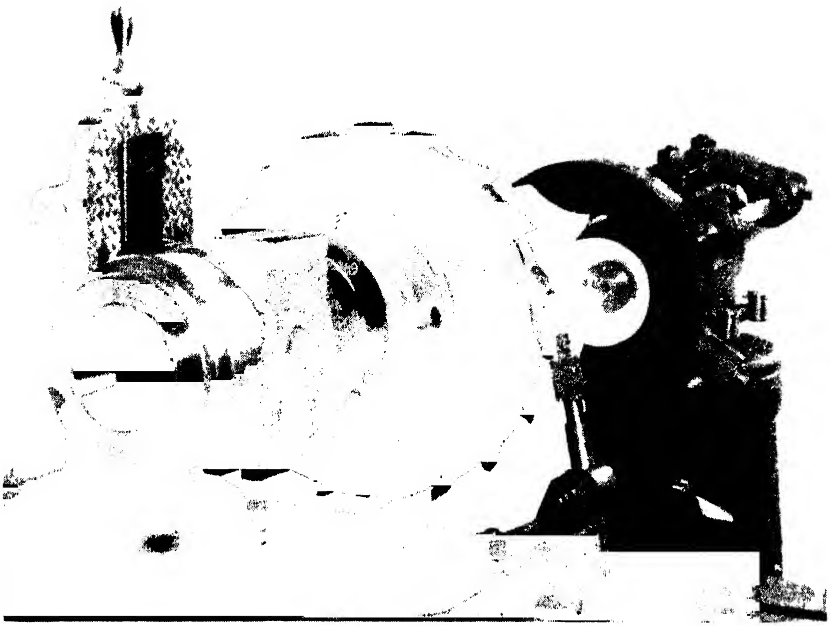


Fig 17. Setup for Sharpening the Faces of a Face Mill
Courtesy of R K LeBlond Machine Tool Co

in Fig. 15. A similar setup may be used for a plain milling cutter mounted on a mandrel and held between centers on the grinding machine table.

When face grinding on any type of carbide-tipped milling cutter is necessary, it should always be done before the primary relief is finished.

Sharpening of Face Mills. Face mills of the type shown in Figs. 2 and 15 are ground in a manner similar to that described for plain milling cutters. For sharpening, the face mill is mounted on a shank just as it would be mounted for work in a milling machine. The mounting should be done carefully so as to assure a true running cutter after the grinding is completed. The face mill is next placed in a universal face mill sharpening attachment such as that shown in Figs. 15, 16, and 17. The operations consist of grinding the peripheral teeth so that they run true, grinding the primary clearance on the cutting edges, grinding the faces

of the teeth and the bevels, and the breaking or chamfering of the corners.

In (A) of Fig. 16 is shown the setup for grinding the periphery of the cutter when old tips are replaced with new ones by means of brazing.

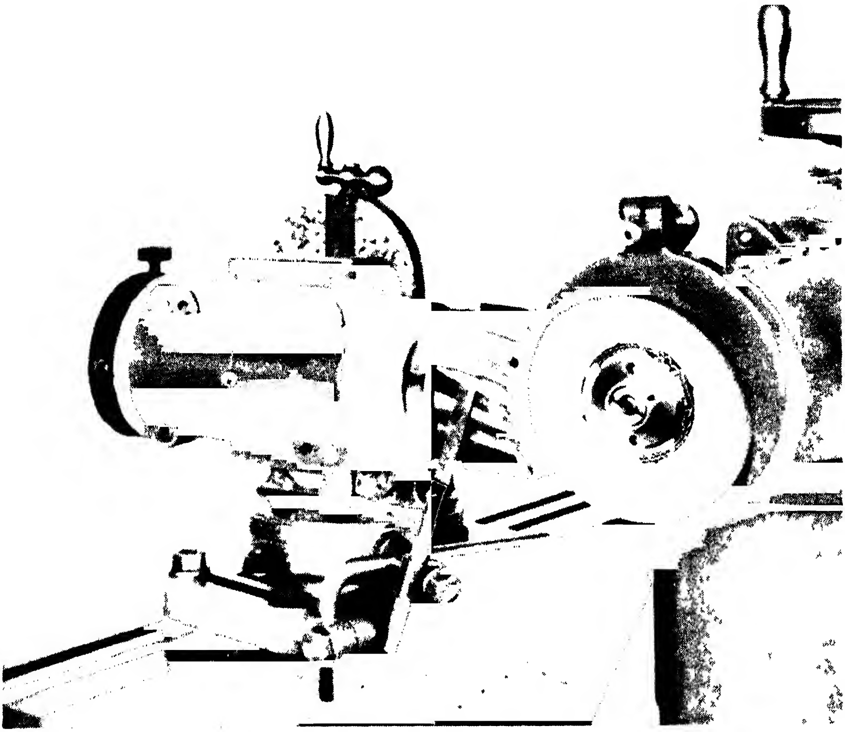


Fig. 18. Sharpening the End Teeth of a Shell-Type Face Mill
Courtesy of Brown & Sharpe Mfg Co

Grinding is done by removing a small amount of material at each pass of the wheel. This serves to keep the carbide from overheating. In this operation, the cutter is rotated by power transmitted through a V belt, hence no tooth rest is used.

In (B) of Fig. 16 is illustrated a setup for grinding the primary relief angles on the periphery of the face mill. Since the face mill shown in the illustration has teeth set at an axial angle, it is necessary to mount the tooth rest on the wheel head of the machine so as to make it easier to keep a constant relief angle on each tooth. For cutting the necessary primary relief, the table with the cutter is raised the required distance, X, which is given in Table III. The rotation of the wheel is such that cutting is done from the edge of the cutter toward its body.

Fig. 15 illustrates the setup for sharpening the teeth on their faces when they become dull, or when the tips are brazed in. In this operation, the tooth rest is located against the back of the tip and on the projection of the carbide beyond the steel body.

Fig. 17 shows a setup for grinding the faces of teeth of a face mill. In this setup, the head is swiveled in a horizontal plane on the base so as to give the cutter a "dish" clearance, after which it is tilted in the vertical plane; that is, the head with the cutter is tilted downward so as to give the necessary relief to the front faces of the teeth. This relief need not be greater than 3° to 5° for many applications, while the swivel is usually about 1°. It is to be noted that the tooth rest supports the tooth which is being ground.

A setup is shown in Fig. 18 for sharpening the teeth of a shell-type end mill or face mill, using a straight wheel. It will be noted that the setup is quite similar to the one shown in Fig. 17.

At the left in Fig. 19 is shown an ingenious mechanism for grinding the ends of the teeth in a face mill. Mounted in front of the wheel is a guide finger which keeps the wheel in contact with the tooth, thus assuring uniformity in grinding the clearance.

When the faces of the teeth, the peripheral, and the front clearances are ground, it is necessary to grind the corners to a chamfer of either 30° or 45°. The setup for grinding the chamfer may be seen at the right in Fig. 19. In this setup, the wheel column is set at 90° to the table. The attachment holding the cutter is set at the base to zero degrees and the table is swung either 30° or 45°, depending on the angle to be used in chamfering the corners. The tooth rest is clamped to the table and is set to support the tooth which is to be ground. The setting is made so that the chamfer will have the necessary clearance of 3° to 5°. To get this clearance, it is usually necessary to give the cutter head a slight tilt while adjusting the tooth rest.

Grinding Procedure for Face Mills. When the cutter becomes dull, it is usually necessary to grind the teeth faces, relief angles, and the chamfer. The amount of material to be removed will depend on how far the cutter was allowed to wear down before being sent to the grinding room.

If the cutter has new teeth brazed in as replacement, it is necessary to circle grind the outside diameter, the faces of the teeth, the face of the cutter, and the corners of the teeth, preferably using a 100 grit diamond wheel of 100 concentration. The teeth are next sharpened individually as explained in the step-by-step procedure that follows:

1. Finish grind the faces of the teeth, using a 220 or 400 grit diamond wheel. The 3 3/4" flaring cup wheel, resinoid bond, is preferred for this job. This wheel is shown in Fig. 20.

2. Rough grind the primary clearance on the periphery of the cutter, using a 100 grit, resinoid-bond diamond wheel. This surface should be ground until all marks are removed. In grinding the clearance, it is advisable to pass from tooth to tooth after each infeed instead of attempt-

Fig. 19. Special Machines for Sharpening Cutters *(Left)*, and Sharpening the Corners of a Large, Face Milling Cutter *(Right)*
Courtesy of Brown & Sharpe Mfg Co

ing to grind each tooth to a full depth or finish. The tooth rest should support the tooth being ground, and grinding should be done from the cutting edge toward the body of the tip.

3. When the cutter is to be used for operations requiring a smoothly finished work surface, the primary clearance on the periphery should be finish ground, preferably using a 400 grit, resinoid-bond diamond wheel of 100 concentration.

4. Rough grind the primary clearance on the periphery of the cutter, using a 100 grit, resinoid-bond diamond wheel. The concavity should be ground to 1° or more.

5. Finish grind the primary clearance on the face of the cutter, using, preferably, a 220 to 400 grit, resinoid-bond diamond wheel

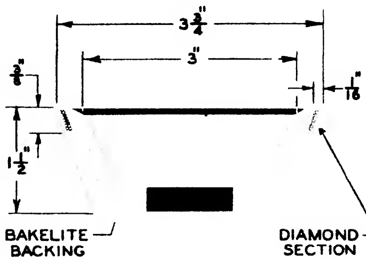


Fig. 20. The Flaring Cup, Resinoid Bond Diamond Wheel

6. Rough grind the chamfer on the cutter with a 100 grit, resinoid-bond diamond wheel and finish with a 220 to 400 grit diamond wheel.

7. If the primary relief land is too wide, it should be reduced by grinding the secondary clearance even before the finish grind of the

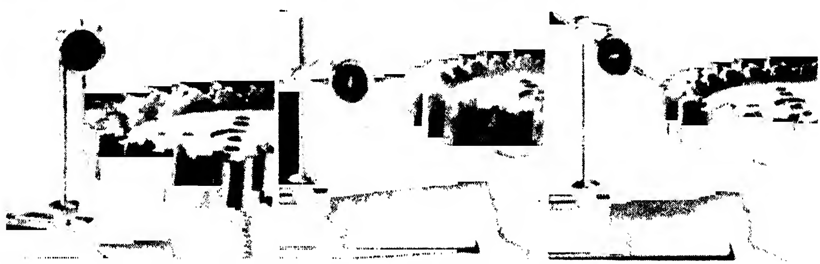


Fig. 21. Testing the Cutter for True Running
Runout on the Face of the Cutter Should Be Checked Carefully on Each Tooth Good Practice Is to Keep Total Runout within the Dial Readings Given in Table V Runout on the Outside Diameter of the Cutter Should Be Checked with the Indicator in the Position Shown Again the Total Should Be within the Values Given in Table V

primary is attempted. A 100 grit silicon carbide or a 100 grit diamond wheel may be used for this operation.

8. Hone the primary clearance land, the face, and the chamfer, using a 400 to 500 grit diamond hone. To keep the hone clean and moist, wipe it off frequently and moisten with kerosene. Care should be exercised not to round off the cutting edges of the teeth. The hone should remove all grinding wheel marks at the points where it is used.

9. Test the cutter for true running, as illustrated in Fig. 21.

10. If the cutter is found to conform to all requirements and passes a finish inspection, it is ready for use.

The procedures explained in the foregoing are also applicable to the sharpening of shell-type end mills. These tools, in reality, are smaller face mills and should be treated as such.

Other Setups and Methods. Fig. 22 shows a setup for grinding the relief behind the cutting edge of a reamer. This is done after the sizing operation, which consists of circle grinding the periphery. In

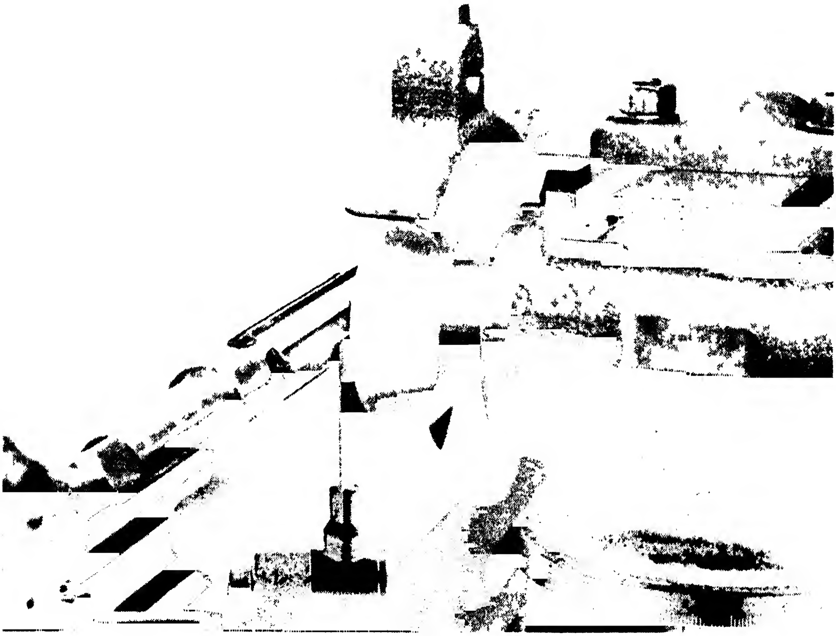


Fig. 22 The Setup for Grinding Hand Reamers
Courtesy of R. K. LeBlond Machine Tool Co.

working with reamers, the circular land is left just a few thousandths of an inch wide. The same setup is used regardless of whether the reamer is carbide tipped or is made of high-speed steel. The only difference in the methods is in the use of wheels of different materials when grinding carbide and high-speed steel. Also, when grinding the carbides, cutting should be done from the edge toward the body of the tooth.

An end mill is a small milling cutter made with a solid tapered shank that fits into a sleeve which, in turn, is held in the spindle of the machine. These tools cut at the end which has teeth tipped with carbide. At the left in Fig. 23 is shown the setup for grinding the peripheral clearance on a small end mill. The setup differs from that given for the sharpening of a face mill or plain milling cutter only in the method of holding the end mill. In this instance, the tooth rest is on the table of the

Fig. 23. Sharpening the Peripheral Teeth of a Two-Lipped End Mill (*Left*), and Sharpening the Face of a Taper Shank End Mill (*Right*)
Courtesy of Brown & Sharpe Mfg Co

machine and is either raised or lowered to give the cutting edge the necessary clearance.

A setup is shown at the right in Fig. 23 for grinding the face teeth of an end mill. This setup differs, again, from that for larger face mills only in the method of holding the cutter. The setup shown in the illustration is used for this type of cutter regardless of whether the cutter is made of high-speed steel or is tipped with sintered or cast carbide.

Tool-Grinding Machines. While it is not the purpose here to discuss the methods of operation of the many tool grinding machines on the market, each of which has special advantages and each of which come with complete instructions regarding its use, it is apropos, nevertheless, to show some of the popular makes of machines used for grinding milling cutters and other multiple-point or edge cutting tools. Thus, it will be possible to become familiar with their appearance and construction.

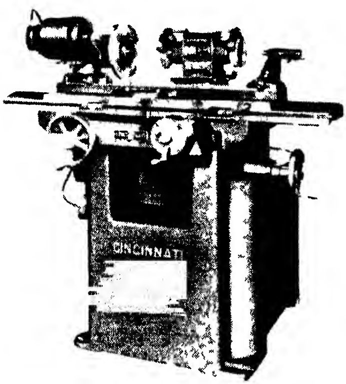


Fig. 24 The Cincinnati Cutter Grinding Machine
Courtesy of Cincinnati Milling and Grinding Machines Inc

A Cincinnati cutter and tool grinding machine is pictured in Fig. 24. This machine finds wide use in grinding multiple-point cutting tools tipped with carbides. Fig. 25 shows the LeBlond cutter grinder which is adaptable for the grinding of all types of multiple-point cutting tools. Fig. 26 illustrates a Landis Tool Co. universal and tool grinding machine which is adaptable for tool grinding and general use.

Grinding Recapitulation. For the purpose of providing a source of ready reference material relating to the grinding of multiple-point tools, the foregoing material has been summarized.

1. To insure high efficiency in production, a keen cutting edge should be kept on the teeth of milling cutters through proper and frequent sharpening.

2. The degree of finish on the cutting edge has a profound influence on the life of the cutter as well as on the finish produced on the work. Therefore, the cutter should be finish sharpened with a fine grit diamond wheel, removing very little material on the final pass.

3. To sharpen the cutter properly, the grinding wheel should be selected with care and mounted so that it runs without vibration and is true on the spindle to within .0005".

4. A properly shaped tootn rest should be selected with a blade length sufficient for the work. However, it should not be too long since the longer blades are less rigid than the short ones.

5. The tooth rest should be placed on the face of the tooth to be ground, about 1/32" away from the cutting edge or just far enough away to clear the grinding wheel.

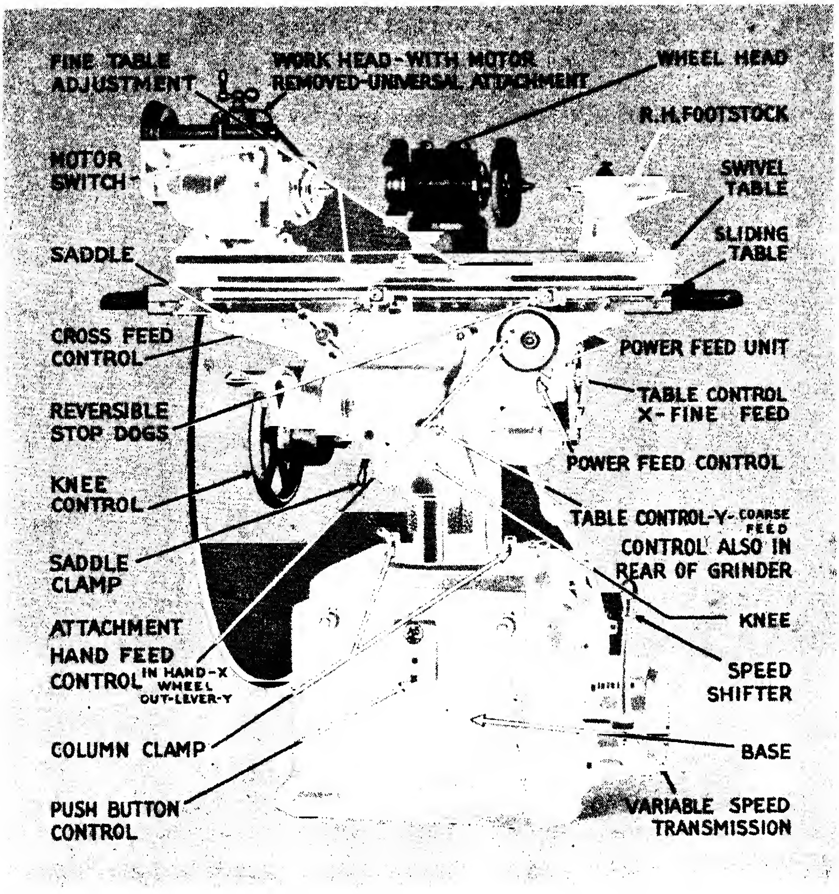


Fig. 25. The LeBlond Cutter Grinder
Courtesy of R. K. LeBlond Machine Tool Co.

6. For roughing cuts, .001" to .002" should be removed per pass. The depth of cut for finishing should be from .0002" to .0003" per pass.

7. In grinding, the carbide should be removed as coolly as is possible. This is done by refraining from forcing the wheel. Heavy cuts will overheat the tips and will cause them to crack. When grinding with diamond wheels, light mineral oil coolants are used.

8. The grinding wheel should be rotated, at least through the finishing phases of the grinding, so that the force of the wheel will tend to

compress the carbide. This will prevent chipping of the cutting edge by the grinding wheel.

9. The grinding wheel should be operated at from 4,000 to 5,000 f.p.m.

10. Safety goggles should be worn at all times during grinding operations.

Fig. 26. The Landis Universal and Tool Grinding Machine, Set Up for Grinding the Teeth of an Angular Cutter
Courtesy of the Landis Tool Company

11. The grinding wheel should be well guarded.

12. The cutter should be inspected for runout after grinding is completed. The permissible runout should not be greater than that shown in Table V.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Into what five groups may the carbide-tipped milling cutters be classified?
2. For what purpose are right- and left-hand side mills used?
3. What is the primary relief of a milling cutter?
4. Does the relief angle have any effect on the finish produced?

5. Is the relief angle greater for machining hard or soft materials?

6. Should the grinding wheel rotate to compress the cutting edge or to put it in tension?

7. What is the function of the tooth rest?

8. Why should the tooth rest support the tooth that is being ground and not some other tooth on the cutter?

9. Why are diamond wheels recommended for sharpening multiple-point cutters?

10. Why should not more than .002" of carbide be removed per pass of the grinding wheel?

11. Why is it desirable to have the milling cutter ground so that it runs true within a definite limit of runout?

12. When grinding with a straight wheel, is it possible to have the clearance greater than that indicated with a protractor?

13. Why is it advantageous to have the clearance angle measured by a drop in dial indicator reading?

14. How much drop on the dial of the indicator per 1/16" is required for the proper primary relief on a cutter which is to be used for machining aluminum?

15. What would be the normal width of the land on cutter teeth for a 10" cutter which is to be used for taking a finishing cut on plastics?

16. For machining semisteel with a 6" face mill, the recommended relief angle behind the cutting edge is 5°. What should the tooth rest setting be for X when the cutter is to be sharpened with a cup wheel?

17. What should the setting be for the cutter in question #16 if it is to be sharpened with a straight grinding wheel which is 8" in diameter?

18. What primary relief is recommended on carbide-tipped cutters for machining white, die cast metal?

19. An 8" mill having 9° primary clearance may be used for machining die cast metal. What should be the setting of the tooth rest when grinding is done with a cup wheel?

20. If the cutter in question #19 is to be sharpened with a straight wheel 5" in diameter, what should the tooth rest setting be?

REFERENCES

Brown & Sharpe Mfg. Co. "Construction and Use of the #10 Cutter and Grinding Machine," Providence, R. I.

Carboloy Company. "Grinding Cemented Carbide Milling Cutters and End Cutting Tools," Engineering Bulletin GT-127.

Carborundum Company. Toolroom Grinding.

Cincinnati Milling and Grinding Machines, Inc. Cutter Sharpening Practice. 1943.

——. Report from Cincinnati Milling. Vol. 1, Nos. 4 and 5, July-August, 1944, and September-October, 1944.

Landis Tool Company. Catalog K-40. (The Landis Universal and Tool Grinder.)

LeBlond, R. K. Machine Tool Co. Tool Room Grinding.

CHAPTER XII

Multiple-Edge Carbide Tools

The Milling Process. It should not be inferred from the material that follows, that any attempt is being made to prove the milling machine superior to other kinds and varieties of machine tools. The relative merits of various machines come only incidentally within the field of this book. However, since this chapter deals with multiple-edged and multiple-pointed tools, it is only natural that the relative advantages and disadvantages of the milling cutter (multiple-edged) should be pointed out as compared with the planer and shaper (single-edged).

In covering this material, therefore, the limiting statements and qualifications should be closely observed in all sections where parallels are drawn. There is no question, on the other hand, that the milling cutter has proven to be one of the greatest advances in the entire history of machining, especially where its use has been found economical, as in all types of repetitive and high-volume production.

Milling Defined. The removal of material, whether it be metal or nonmetallic, by means of a rotary cutter having teeth on its periphery, is called milling. This work may be done with a single tool held in a suitable holder, which in turn may be held in the spindle of the machine and rotated by it. Milling may also be done by a number of tools held in the periphery of a disc and rotated by it. Such tools in the periphery of a disc are called teeth. The multiple-cutting-edge milling cutter may be held in the spindle of the machine or on a suitable arbor, and rotated so as to cut the work fed against it or toward it, but not necessarily in the same direction as the rotation.

Milling is said to have originated among Swiss clock and watchmakers as far back as the sixteenth century. However, the milling machines of that day were quite crude. The cutters had fine teeth, so fine in fact that they bore little resemblance to the cutters of today, and even less to the carbide-tipped cutters that will be discussed further in the chapter. Eli Whitney of cotton gin fame is generally credited with inventing, or at least with introducing, the milling machine in more or less modern form in America.

The nineteenth century, however, actually witnessed the real development in the art of milling. This resulted from the introduction of the universal milling machine, of suitable tool grinding machines, and grinding wheels with which the cutters could be resharpened. These were the

important steps forward because they made possible the economical manufacture and resharpening of milling cutters. The introduction of the universal milling machine made possible a greater adaptability of milling to many jobs that had previously been done on other machines. This development enlarged the field of milling to such an extent that today the milling machine is almost a necessity in even the smallest production and tool plant.

A great contributing factor to the art of milling was the introduction of high-speed steel for the construction of milling cutters. This new material made possible the increase of cutting speeds by at least 100 per cent over the previously used carbon-steel cutters, and allowed a comparable increase in feeds and depths of cut. This innovation immediately called for heavier and more sturdy machines, powered with motors of greater horsepower. Such machines were thereupon developed by the manufacturers.

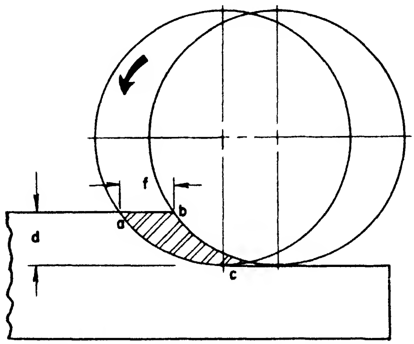


Fig. 1. Variation in Chip Thickness in Milling

The latest great step forward in the art of machining by milling was the introduction of milling cutters tipped first with the cast alloys, and later with the sintered carbides. This made possible the trebling and quadrupling of the speeds generally employed with the high-speed cutters and enabled cutting with greater feed per tooth than was ever before dreamed possible. Again, the introduction of negative rake cutters made possible additional increases in cutting speeds and feeds. These developments called for still more rigidity in the machine, greater rigidity of the tool and tool holding devices, and more rigid work holding fixtures. Accompanying these factors, of course, was the demand for still greater power to be available at the cutter. Such are the considerations that today influence the design of milling cutters, of the machines, and of the work holding fixtures.

Whether the work is done by milling, turning, shaping, or planing, cutting action with a sharp-edged cutting tool is governed by the same basic laws insofar as the actual separation of the chip material from the parent metal is concerned. In detail, however, the cutting action of the milling cutter is different from that of the turning tool. In turning, a continuous chip usually is removed from the material being worked, while in milling, the chip varies in thickness. This is shown in Fig. 1 where the milling tooth takes a cut of thickness a-b, which is equal to the feed, f. The cutter disengages the work at c with no cut at all. Unless another tooth comes into action, the cut is terminated and there will then be no cutting done until the tooth comes around again to engage the work. This is the cutting action of a single-tooth milling cutter

But, whether the cutter has one or more teeth, the cutting action of each tooth is intermittent.

In the process of cutting by milling, a maximum force is required to make the cut at a-b, and the minimum effort at c, whether the cutter enters the work at c or leaves it there. Thus, not only is the cutting action intermittent, but the intensity of the cutting force on each tooth varies with a maximum to a minimum in each cut. These are the factors

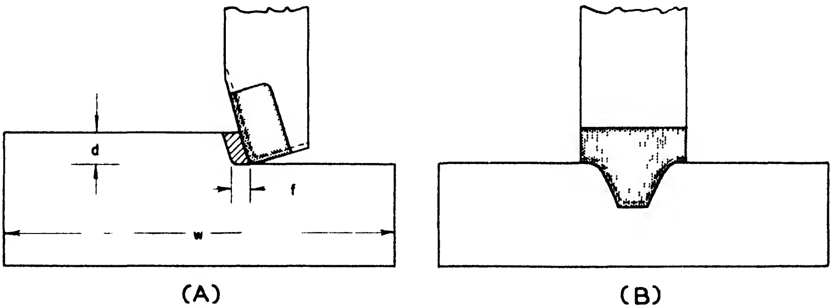


Fig. 2. Machining with Single-Point Tools in the Planer or Shaper, Taking a Plain Straight Cut (A) and a Formed Cut (B)

which must be taken into consideration in designing cutters, cutter holders, and work holding devices or fixtures.

Classification of Milling. Cutting operations that are readily grouped under the general term of milling are plain milling, side milling, angular milling, grooving or slotting, and form milling.

Plain milling is defined as the performance of a cutting operation on a plane surface which has elements parallel to the axis of rotation of the cutter. It is usually done with a plain milling cutter or a slab mill.

Side milling refers to the cutting operation performed on plane surfaces that are perpendicular to the axis of rotation of the cutter. Side milling cutters or face mills and end mills are used for these operations.

Angular milling refers to the ma^hining of plain surfaces that are inclined to the axis of rotation of the milling cutter. Angular cutters are used in these operations. Sometimes, however, the work is held in an angular relation to the cutter, in which case a plain milling cutter or a face mill is used.

Grooving of slotting refers in general to the cutting of grooves in such work as keyways, splines, etc. The grooves may be rectangular, have inclined sides, or be formed.

Form milling refers to machining with rotary cutters where the surfaces machined are not plane but are curved. The surfaces have the same profile throughout the direction in which the work is fed against the cutter. In other words, the elements of the surface are parallel to the line of motion of the work in the direction of the cut.

Milling versus Other Methods. The machining of plain, angular, or formed surfaces may be done, and frequently is done, in the shaper or in the planer. Plain and angular surfaces may thus be machined with a single-point tool as shown in (A) of Fig. 2, or they may be machined with a multiple-point rotating tool in the milling machine. Formed surfaces of the type shown in (B) of Fig. 2 may be machined with a single-point forming tool held either in the shaper or planer, or by a multiple-point tool. In the latter instance, the job again becomes milling.

Since the work can be done equally well by either method—that is, by shaping, planing, or milling—one must look into the relative advantages and economy of these methods in order to make a comparison. In shops where milling machines are not available, there may be no choice in the method of machining. The method is there dictated by the availability of the machines and the tools. When milling machines are available, the method of machining is decided by the economy of the method and by the tool available for doing the work.

The relative merits of milling and shaping machines can be readily seen by comparison if it is assumed that a piece of work similar to that shown in Fig. 2 is to be machined. The dimensions of the workpiece are 6''×12''. A 1/8'' depth of cut is to be taken from the surface. The work is to be clamped on the milling machine table against an angle plate and suitable stops, using bolts and clamps. The work would be held in the shaper in a similar manner. From the description of the conditions, it may be assumed that the setup in the milling machine will consume more time than that in the shaper. For the purpose of the comparison, it can be assumed that 15 minutes will be required for the shaper setup, whereas 45 minutes will be required for the same setup in the milling machine. Thus, there is a definite advantage in the setup time in favor of the shaper. However, it is in the cutting time and in the subsequent setting of work in the machine when more pieces are to be machined that the advantage may shift from one machine to another.

Using a negative back rake on a single-point, sintered-carbide tool mounted in the shaping machine, the speed would be about 30 strokes per minute and the feed 1/32'' per cutting stroke. The time required to take the cut would then be equal to

$$\frac{6 + 1/8}{30 \times 1/32} = 6.54 \text{ minutes}$$

(The addition of the 1/8'' is for the approach of the tool to the work.) In shaping operations, it is usually necessary to take two cuts, one for roughing and the second for finishing. Should this be the case, the time for machining one piece would then be 13.08 minutes. The total time would then be 28.08 minutes since the 15 minute setup time must be added to the time consumed in machining the piece.

If an 8'' face milling cutter having 10 teeth is available for this job and if there is sufficient power to be had so that the machine can be operated at a speed of 600 f.p.m. with a feed of .006'' per tooth, the time required for machining will be much less than when the shaper was used. Using a formula which is developed in detail in Chapter XIII, it is found that the time required in the milling machine would be but 0.78 minutes per piece. If the setup for the first piece takes 45 minutes as was stated, then the total time for milling one piece would be 45.78 minutes as against 28.08 minutes required for the same job in the shaper. In milling, because the tool has multiple cutting edges and the feed is finer, one cut is usually sufficient to finish a surface of this sort.

Comparing the results just given, it is evident that the advantage thus far rests with the shaper method of machining. However, assuming there are 50 pieces to be machined, and that for shaping the setup time will be four minutes per workpiece after the initial setup of 15 minutes for the first piece is made, and that for milling it will be 10 minutes after the first setup of tools, the total time for 50 pieces by shaping and by milling under the same conditions will be as follows:

SHAPING		MILLING	
Initial setup of one piece	15 min.	Initial setup for one piece	45 min.
Setup of 49 pieces at four minutes	196 "	Setup for 49 pieces at ten minutes	490 "
Cutting time for roughing only	327 "	Cutting time for 50 pieces	39 "
Setup time for finishing 50 pieces	200 "	Total time	574 min.
Cutting time for finishing	327 "	or 9 hours,	34 min.
Total time	1,065 min.		
or 17 hours,	40 min.		

Thus, the time required for milling is only a little more than half that required for shaping, thereby allowing the milling method to win hands down. The time would be still less if proper fixtures were available for setting the workpiece quickly in place in the milling machine. If the setup time for the two machines should be the same, the advantage will be with the milling machine even on limited number of piéces to be machined.

Disregarding other factors, such as the cost of the machines, of fixtures, and cutting tools, it is apparent that milling, whenever it can be applied to a number of pieces, is more economical than shaping. It is for this reason that milling has displaced shaping to a great extent in volume repetitive production. If a planer machine were available and the same number of pieces considered in the foregoing material were placed on its platen or table, the production would be accelerated some-

what, but it still would in no way be equal to that of the milling machine.

Form Milling. Formed work, such as shown in (B) of Fig. 2, can also be machined in the shaper, planer, or milling machine. To machine

Fig. 3. Typical Setup in a Horizontal Milling Machine
Courtesy of Brown & Sharpe Mfg Co

it in the shaper, it would be necessary first to rough the work nearly to the desired shape with the roughing tool, then finish it with a finishing tool. The tool would feed downward at 0.002'' to possibly 0.006'' per stroke. In milling, on the other hand, only one cut would be necessary and the shape of the cutter would be reproduced in one pass. Here, the cost of the cutter and the quantity of work to be done would have to be taken into consideration. The single-point cutting tool is low in cost while the formed type milling cutter is relatively expensive. Therefore, the decision on the method of machining probably would depend on the cost of the tools per piece of work done.

It is apparent from the foregoing that the question of milling as against shaping depends primarily on

1. The number of pieces to be machined
2. The size of the piece parts
3. The machines available to do the work
4. The tools available for the job
5. The cost of the tools and the work holding fixtures

Milling Machines. Milling machines are machine tools which remove material as the work is fed against a rotating cutter. Fig. 3 illustrates in a general way, the principle of cutting and the setup of the work. In this machine, the cutter has only one motion which is rotation. The cutter, as is seen in the illustration, has a series of cutting edges on its circumference. Each one of these cutting edges acts as an individual, single-point cutting tool during its cycle of rotation. The work to be done in the milling machine is mounted in a fixture which is fastened to the table. The work may be fed against the cutter either longitudinally, transversely, or vertically. The table may have a rotational movement in some milling machines, but these are special production tools.

The milling machine is nearly as versatile as the lathe. Either flat or formed surfaces can be machined to an excellent finish and a high degree of accuracy with it. In addition to flat and formed surfaces, the milling machine can produce slots, gear teeth, cams, etc. Drilling, reaming, counterboring, countersinking, and even turning, boring, and facing can be done in a milling machine. In short, most operations that can be done on a planer, shaper, gear cutter, broaching machine, or drill press can be done on the milling machine. The advantages of a milling machine over other machines can be summed up as follows:

Milling cutters are efficient tools and can be used for a long time before regrinding is necessary.

Heavy cuts can be taken without sacrificing the finish produced and/or the dimensional accuracy.

The work is usually finished in one pass of the table.

The work is usually produced much faster than can be done either in the shaper or planer, once a setup has been made.

These characteristics of the machine, its ability to do not only milling but also drilling, reaming, counterboring, countersinking, and even short turning, together with the wide variety of cutters that can be used in it, make the milling machine indispensable in most shops and a necessity in every tool room.

Classification of Milling Machines. Milling machines are made in a great variety of types and sizes. Their design is such that it is difficult to classify them. Some machines have hand feeds, others have mechanical feeds, and still others have hydraulic feeds. A general classification, however, is as follows:

Column and knee-type milling machine which may be either a plain or a universal type

Vertical milling machine with either a rotating or a reciprocating table

Planer-type milling machine

Fixed-bed milling machine

Special milling machine

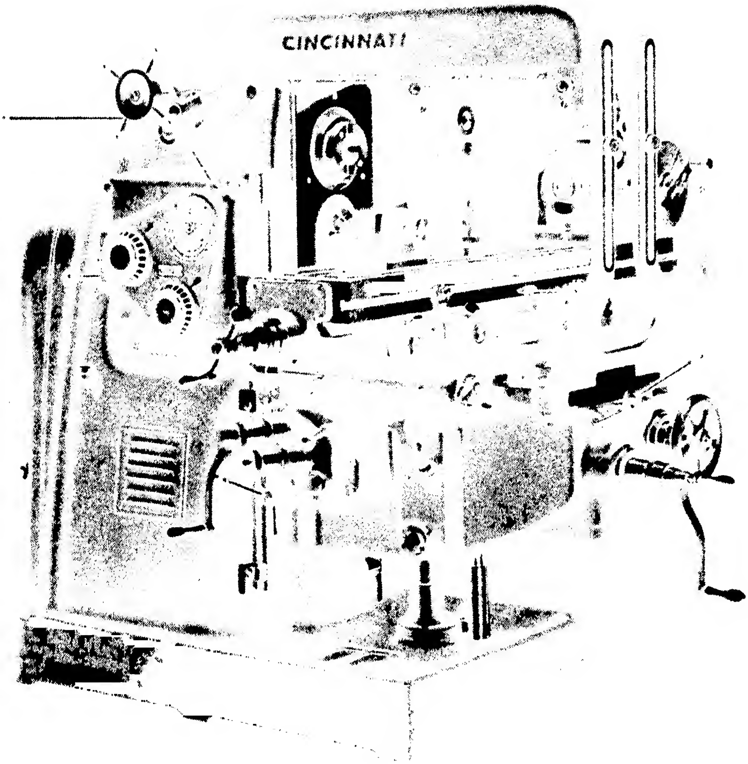


Fig 4 Universal Milling Machine of the Knee Type
Courtesy of Cincinnati Milling and Grinding Machines Inc

The plain milling machine of the knee type was illustrated in Chapter II. This machine is used extensively for general work in the toolroom or the production shop.

The universal milling machine shown in Fig. 4 is the most versatile of all types. It is essentially a toolroom machine and is constructed for extremely accurate work. In appearance, it is similar to the plain knee type shown earlier but differs from it in two respects. The work table can swivel to an angle in the horizontal position and the machine is provided with an index or dividing head which can be connected by gears to the lead screw of the machine table. This permits the cutting of helixes and spirals as are found in gears, drills, milling cutters, reamers, cams, etc. These machines are usually provided with a vertical milling attachment, rotary table, and other accessories.

In Fig. 5 is illustrated the vertical type of milling machine. The spindle is in the vertical position. Some machines are built with a spindle head that can be swiveled. Others have spindles fixed in the vertical direction. The machines are usually provided with a short axial movement of the spindle to permit down feed. Cutters used in this type of machine are of the end-mill type. Such machines are particularly adaptable to making facing cuts, finishing recesses, profiling, and die sinking.

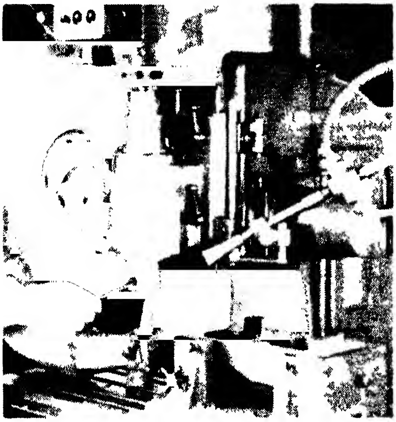


Fig. 5. Typical Vertical Milling Machine
Courtesy of Brown & Sharpe Mfg Co

The planer-type milling machine is shown in Fig. 6. In this machine, the work is carried on a long table having only longitudinal movement, similar to that of the planer table, the work being fed against the cutter at the proper speed. Transverse and vertical movements for adjusting the depth of cut are provided on the cutter spindle carriers. The planer-type milling machine is essentially a high rate production machine.

A fixed-bed type machine was shown at work in Chapter I. This type of machine is strictly a production tool and is of very rugged construction. Such machines are called "simplex," "duplex," and "triplex," indicating the number of spindles with which the machine is equipped. The machine tool shown in Chapter I was a duplex machine, used to finish both sides of a workpiece at the same time.

Special milling machines have been developed to take care of special types of milling operations. There are special machines for thread milling, for profiling, and for cam milling. There are drum-type machines which permit continuous loading and unloading of the work while the machine is in operation.

Economics of Milling. Cutting speed is defined in Chapter XIII as being the lineal, peripheral speed resulting from rotation and is the product of the circumference of the cutter and the number of revolutions per minute that it makes. In practice, the cutting speed will depend on a number of variable factors which will be developed more fully in the next chapter. These include:

1. The kind of material being cut, its hardness, toughness, and abrasive character
2. The amount of material to be cut
3. The material of which the cutter is made
4. The power available at the machine spindle
5. The speed and feed range of the machine

6. The rigidity of the machine, the arbor, the cutter, the work itself, and the work holding fixture
7. The relation between the depth of cut to be taken and the feed
8. The finish desired
9. The cutting fluid used

Normally, the cutting speed and feed for the material being worked

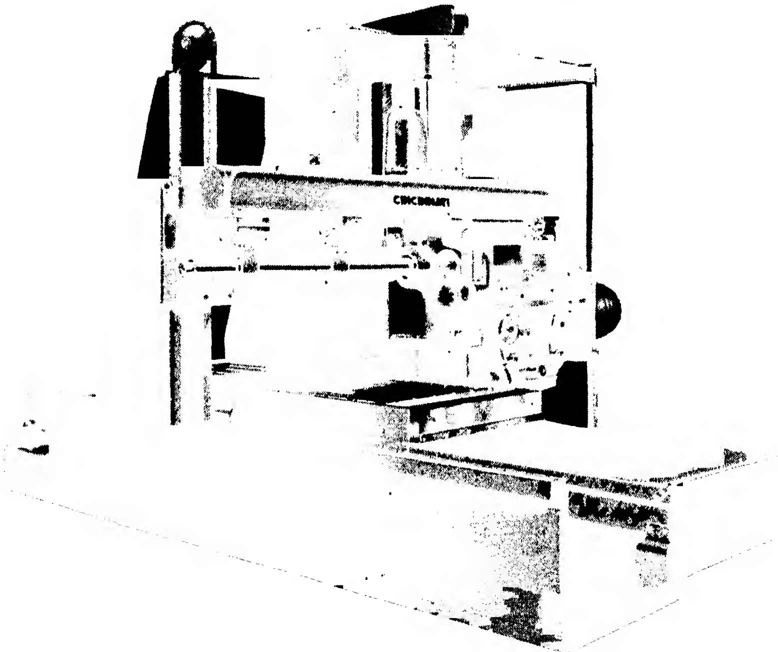


Fig 6 A Planer-Type Milling Machine
Courtesy of Cincinnati Milling and Grinding Machines Inc

should be as high as possible, while being commensurate with economical tool life between grinds, finish obtained, and the resulting dimensional accuracy of the work.

In order to keep the tool working for a reasonable length of time, the harder the material is, the slower should be the cutting speed. Very hard materials cannot be milled successfully save with cutters tipped with the carbides. These cutters are able to resist abrasion and operate more efficiently than high-speed steel cutters.

Economical speeds for high-speed steel and sintered carbide cutters are given in Table I, and may be used for comparison and reference. In general, sintered-carbide milling cutters operate at a speed from three to four times greater than that recommended for the single-point tool used for turning, facing, and boring. This is because the cutter tooth

TABLE I. CUTTING SPEEDS WHEN MILLING WITH HIGH-SPEED STEEL AND SINTERED CARBIDE CUTTERS

Material Cut	Cutting Speeds in Feet per Minute	
	High-Speed Steel	Sintered Carbides
Aluminum and magnesium..	500-1,000	1,000-20,000
Bakelite..........................	100- 200	1,000- 1,500
Brass.............................	100- 200	400- 1,000
Bronze	30- 200	300- 800
Cast iron	50- 120	400- 600
Cast iron, malleable.........	80- 100	500- 800
Copper	100- 200	600- 1,500
Monel metal	70- 80	350- 500
Steel:		
Alloy, heat-treated.........	30- 50	250- 600
Alloy, not heat-treated	50- 70	300- 650
High-carbon, annealed	50- 75	450- 750
Low-carbon, cold drawn ..	60- 100	600- 1,200
Stainless	60- 100	450- 750

engages the work intermittently, and, turning in the air, dissipates the heat acquired during the cutting.

The feeds used in milling with high-speed steel and with carbide cutters are not materially different. Sintered carbide cutters should not be used for very fine feeds of less than 0.002" per tooth, because at lower feeds there seems to be an excessive amount of wear on the cutting edges from the abrasive action of the material being cut.

If we take as an example, the milling of drop forgings, heat treated to 300 Brinell hardness, which is within the machining range of high-speed steel or carbide cutters, it can be shown that milling with carbide is faster and more economical than performing the same operation with tools of high-speed steel. In machining a material of 300 Brinell hardness, the cutting speed for high-speed steel tools may be from 30 to 40 f.p.m. while for the sintered carbide cutters it will be from 400 to 500 f.p.m. Assuming that two cutters of the same size are available for the job, one being of high-speed steel, 6" in diameter, and having 12 teeth, the other being of sintered carbide, also 6" in diameter, but having only 8 teeth; and that the feed in either case will be 0.004" per tooth, let us determine which cutter will machine more material in one minute of time.

At a cutting speed of 40 f.p.m., the high-speed steel cutter will make 25.4 r.p.m. At a cutting speed of 500 f.p.m., the carbide cutter will make 318 r.p.m. The length of cut in one minute will be the product of the chip load or the feed per tooth, the number of revolutions per minute

the cutter makes, and the number of teeth in the cutter. Thus, the length cut by the high-speed steel cutter is

$$0.004 \times 12 \times 25.4 = 1.22''$$

For the sintered carbide cutter, the length of the cut would be

$$0.004 \times 8 \times 318 = 10.2''$$

These results indicate that the sintered carbide cutter has an advantage in cutting in the ratio of 1.22 to 10.2 or 1 to 8.3. In other words, the sintered carbide cutter will machine the work 8.3 times faster than the high-speed steel cutter.

In per cent, the performance of the sintered carbide cutter is even more outstanding.

$$\frac{10.2 - 1.22}{1.22} \times 100 = 745 \text{ per cent}$$

It is obvious that the carbide-tipped cutter should be used if the quantity of work is large enough to warrant the additional investment.

The cost of milling cutters suitable for this kind of work would be about 30 per cent higher for sintered carbide. This difference in cost is not great when there is sufficient work on hand or in sight, for the investment in carbide-tipped milling cutters will quickly pay for itself.

Important factors which should be considered in milling with sintered carbide cutters are whether the machine available is capable of the higher speed required for the use of carbide, and whether the horsepower necessary to take the cut contemplated is sufficient. Because of its greater speed, the power requirements for the cutter tipped with carbide will, in general, be greater than that for the high-speed steel cutter. Computations of cubic inches removed by the cutter per minute and the horsepower required to remove such material are discussed in detail in Chapter XIII.

Fig. 7. Setup for Milling a Dovetail Slide
Courtesy of Brown & Sharpe Mfg. Co.

Fig. 7 illustrates a typical setup for milling dovetail slides made of a fairly hard cast iron. The setup was made in a Brown & Sharpe vertical milling machine using a high-speed steel cutter. The cutter was

TABLE II. PRODUCTION FIGURES FOR SPECIFIC JOB USING TOOLS OF HIGH-SPEED STEEL AND CARBIDE

Speed, Feed, and Production Data	High-Speed Steel Cutter	Sintered Carbide Cutter
Speed in r.p.m.	100	250
Feed in inches per minute	3	10
Depth of cut	1/16	1/16
Number of pieces per grind	80	640
Pieces produced per 8-hour day	60	180

later changed to one tipped with sintered carbide. The data obtained is presented in Table II and shows interesting results.

Thus, it can be seen that the increase in production was

$$\frac{180 - 60}{60} \times 100 = 200 \text{ per cent}$$

Another example of economical production in milling was shown in the milling of cast-iron valve bodies. Data collected on the tool performances is given in Table III.

TABLE III. PRODUCTION FIGURES FOR SPECIFIC JOB USING TOOLS OF HIGH-SPEED STEEL AND CARBIDE

Speed, Feed, and Production Data	High-Speed Steel Cutter	Sintered Carbide Cutter
Speed in f.p.m.	63	265
Feed in inches per minute	2.6	12.50
Depth of cut per side	1/8	1/8
Pieces per grind	290	2038
Output per hour	6	30

On this particular job, $53.00 was saved every eight hours through using cutters tipped with sintered carbide.

Limitations in Milling with Carbide. The preceding paragraphs have pointed out that milling with cutters tipped with sintered carbides is a necessity when the material to be cut is hard, when the quantity of work to be done is large, or when the work has to be done quickly. One of the illustrative examples also showed that the increase

in production amounted to some 745 per cent. There are instances where milling with sintered carbide tipped cutters gave increased production over the high-speed steel cutters in the ratio of 20 to 1. It has even been higher in other cases. A notable example of this is the milling of airplane structural parts made of aluminum at cutting speeds from 15,000 to 20,000 f.p.m., a speed previously unheard of for cutting tools. In this case, the cutter had only two teeth, and the chip load was 0.002'', the distance the work moved past the cutter in one minute, the cutter making 15,000 r.p.m. is equal to

$$0.002 \times 15{,}000 \times 2 = 60''$$

as against 6'' for a high-speed milling cutter of the same size, having four teeth, taking the same chip load, and operating at a cutting speed of 2,000 f.p.m.

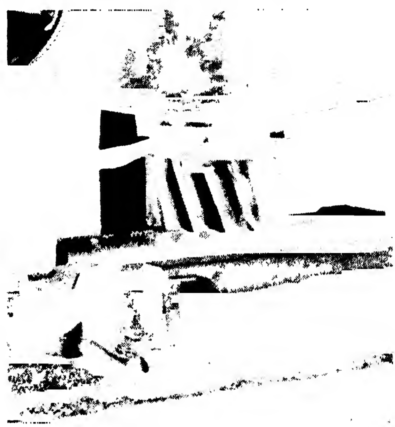


Fig. 8. A Typical Face Milling Operation
Courtesy of Brown & Sharpe Mfg. Co.

The additional speed at which sintered carbide cutters must operate, if efficient, calls for greater power at the machine spindle, and more rigidity in the machine, in the work, and in the work holding fixture. Since this additional power is not always built into the machine, the full ability of the carbide cutters cannot always be realized.

Face Milling Cutters. Thus far, the greatest developments in sintered carbide cutters have taken place in the type used for facing operations. A cutter of this type is shown in Fig. 8. The art of milling has been greatly enhanced in this field because, in the facing operation, relatively small depths of cut are usually taken at reasonable feeds. This naturally requires less power for the cutting done than in other types of milling. Again, in face milling with carbide-tipped teeth, the cutter body is of heavy construction. This provides a flywheel action during the cutting, eliminating much chatter and making the intermittent action of the cutter more smooth. In short, face-milling cutters with teeth tipped with carbides have been developed to a fairly high degree of perfection.

Plain and Side Milling Cutters. The plain and stagger-tooth milling cutters shown in Chapter X are next in line of progressive development. These cutters are used for milling grooves, for facing work, and for straddle milling on two sides of the work as shown in Fig. 9.

This type of milling may be limited by lack of sufficient rigidity in the milling machine arbor which holds the cutters and rotates them against the work as shown in Fig. 9. Its great advantage lies in the

ability of these cutters to machine hard materials and even steel which has been heat-treated to relatively high degrees of hardness. For cutting hard materials, the teeth should have a negative radial rake angle of 5° to 15°.

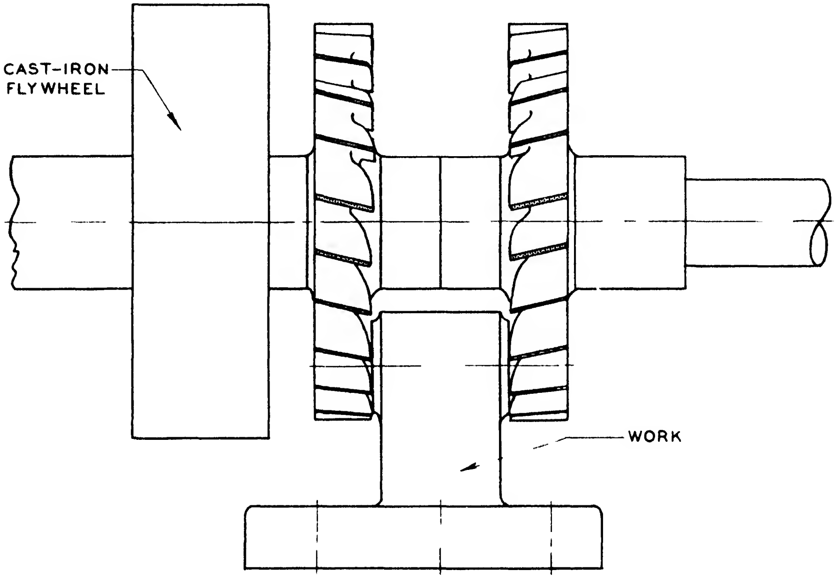


Fig. 9. Straddle Milling, Using a Cast-Iron Flywheel

For smooth cutting action in straddle milling with carbide cutters, a flywheel may be mounted on the milling machine arbor. The flywheel should be mounted as close to the cutter as possible but not so close as to interfere with the cutting operation. How such a flywheel may be applied is shown in Fig. 9.

Slab Mills. Less development has taken place in the milling of flat surfaces with a slab mill, an operation shown in Fig. 10. This particular field has not been explored to any great extent although many "plain" cutters in common use are, in reality, narrow slab mills.

The principal reason for the slight application of carbide tipped cutters to slab milling is the lack of rigidity in the arbor which must be used to mount the cutter. When the cutter takes a heavy cut, it creates deflection in the arbor which causes it to bind in the support bearing because of overheating. Slab milling awaits development in the design of more rigid machines; bearings cooled, perhaps by some circulating liquid; methods of holding the cutter more rigidly than is now possible; and, perhaps, even in the design of the cutter itself. Another reason for the lack of application of carbide cutters to slab milling is the

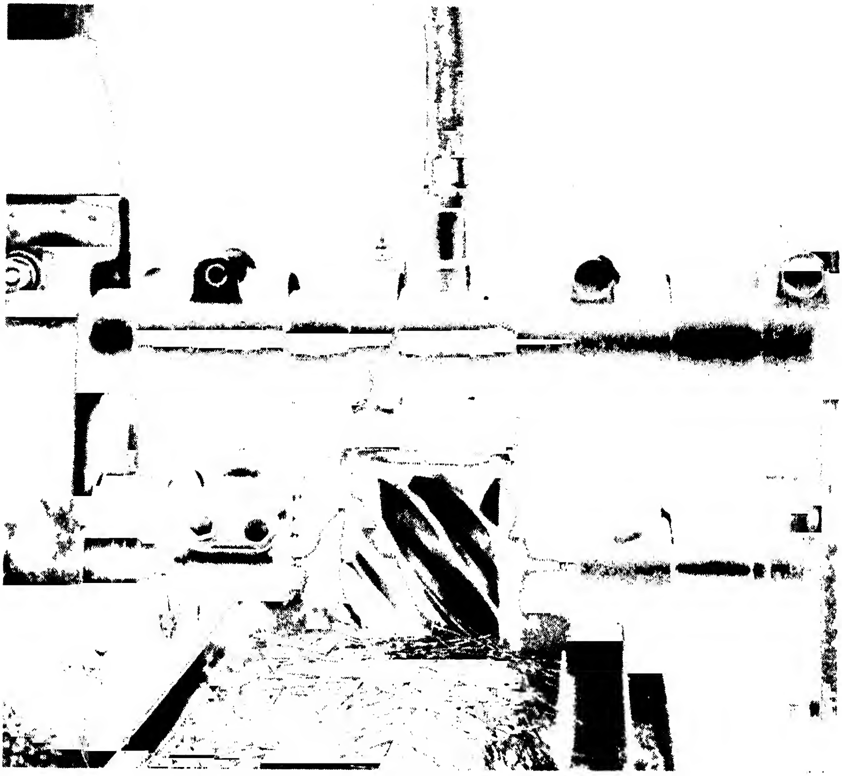


Fig. 10. The Slab Milling Operation
Courtesy of Brown & Sharpe Mfg Co

difficulty experienced in making the helical teeth shown on the high-speed steel cutter shown in Fig. 10. Attempts have been made to work around this difficulty by means of a spiral type of cutter such as is illustrated in Fig. 11. It is, in reality, a broad slab mill with teeth staggered so that at no time is a very wide cut taken. Since a rather narrow cut is made, the power requirements are not great. This cutter also awaits further improvements and developments.

End Mills. End mills are similar in design and action to face mills. These tools, as the name suggests, cut with the end and also with the sides. They are well suited for cutting slots that do not lend themselves to cutting with a plain milling cutter of the type shown in Fig. 12. Such slots often end in the material and are required to be the same depth throughout their length. Through slots also can be milled with end mills.

One type of slot milling is demonstrated in Fig. 12. Using this

method of milling, fairly hard castings were machined with high-speed steel cutters. When cutters tipped with sintered carbide were substituted, the speed was increased from 60 to 200 f.p.m. and the output for eight hours of work was doubled, giving an increase in production of 100 per cent. Tool life also was substantially increased in this instance, 90 pieces per grind being the limit with high-speed steel, but 800 per grind when the carbide tipped mill was put to work. This resulted in the saving of additional time.

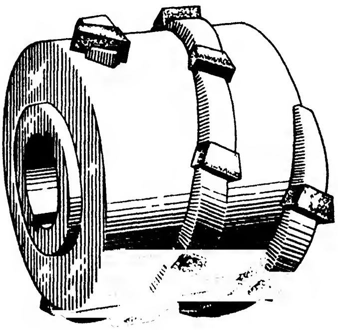


Fig. 11. A Double Spiral "Slab" Mill

This type of cutter has limitations in that the tool with a small diameter cannot be operated at high efficiency because of the low cutting speed at which it may run. Most ordinary milling machine spindles are not capable of very high speeds. Nevertheless, wherever hard materials are to be machined or the quantity of work is sufficient to warrant the additional investment in carbide tipped end mills, the investment will pay dividends in greater output per man hour, in greater tool life between grinds, and in better finished products.

Carbide-Tipped Reamers. Reamers are multiple-edged tools cutting with side edges, and are used for enlarging holes previously drilled or bored. The material removed by reaming should be from 0.005" for reamers of small diameters, to 0.015" for reamers from 3/4" diameter and up. Where the accuracy of the hole to be finished is of prime importance, the reamer should remove even smaller amounts of material. However, the amount should not be so small as to result in a discontinuous cutting action. In many operations where a particular fit is desired, reamers must produce holes to a fraction of a thousandth of an inch! A typical carbide tipped reamer is shown in Fig. 13.

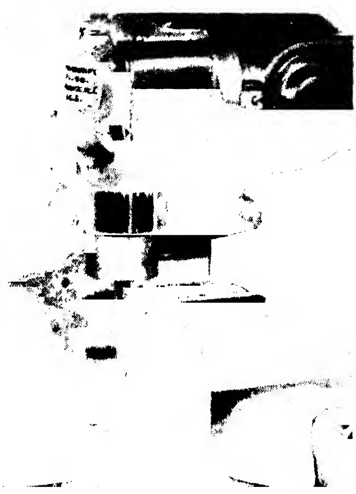


Fig. 12. Slot Milling on a Vertical Machine with an End Milling Cutter
Courtesy of Brown & Sharpe Mfg. Co.

In use, the reamer should be held so that its center line coincides with the center line of the work. Where this alignment does not exist, trouble may be expected. Misalignment will generally occur from careless setting of reamers or from worn-out or dirty reamer holding devices.

When the reamer is held rigidly in the machine, misalignment may cause poorly finished and oversized holes. If the axis of the reamer does not coincide with the axis of the machine and is at an angle to the work, as shown by the exaggerated drawing in Fig. 14, the cutting end of the reamer will be forced into the work, and, as it is advanced, the cutting action will be confined to one side of the reamer. This will result in a bell-mouthed hole. Excessive pressure will be applied on one side of the reamer as a result of this uneven cutting action. This can easily cause gouging and tearing of the reamed surface and may also result in damage to the flutes of the reamer. It is therefore necessary in reaming to have good alignment of the tool with the work.

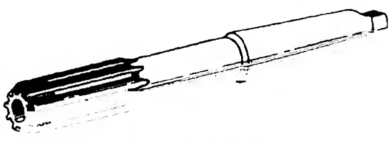


Fig. 13. A Typical Carbide-Tipped Reamer

To eliminate oversize, torn, or bell-mouthed holes, means must be found to neutralize the effect of misalignment. This can be done by correcting conditions on the machine and the tool holder. The usual method of accomplishing this is by cleaning the holders, replacing worn-out sleeves and bushings, and relocating the holders.

Another method of correcting misalignment is to use floating holders for reamers. There are several types and designs of floating holders. Each one has definite advantages. The general appearance of a floating holder for a reamer is illustrated in Fig. 15. It consists of a sleeve, A, into which the reamer shank, B, fits loosely, and which is kept from

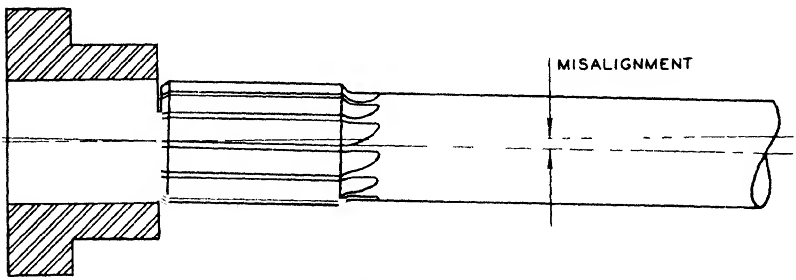


Fig. 14. Misalignment of the Reamer with the Work Must Be Avoided

rotating by a pin, C, which passes through the sleeve and the reamer shank. If the pin fits snugly in the reamer shank and in the sleeve, the adjustment of the reamer for self-alignment is only in the up and down direction, or the rotation about pin C as an axis. On the other hand, if the pin fits fairly loosely in the shank of the reamer, the reamer is capable of some axial adjustment in the horizontal and the vertical planes and its action becomes somewhat more universal. In such cases, the sleeve should not be made with the hole too large. If the hole is too large, the reamer will deflect down from the horizontal line and will in-

convenience the operator in locating it in the hole to be reamed. If the misalignment is so great that excessive flotation is necessary, the condition should be corrected before the reamer is used. A well-designed floating holder as applied to turret lathe practice is shown in Fig. 16.

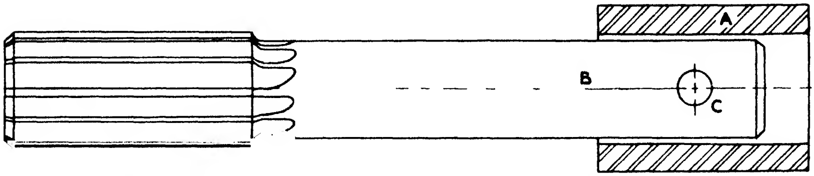


Fig. 15. A Floating Holder for Reamers Will Assist in Making the Proper Alignment

Carbide Grades for Reamers. Grades of carbides commonly used for reamers are usually the same as those used for milling cutters. For cutting nonferrous materials and soft steels, carbides are used that are suitable also for cutting cast iron. These are found to perform satisfactorily. The steel cutting grades of carbides will be found more satisfactory for reaming tough alloy steels. In reaming with carbide tipped reamers, the work can be done at high speed with long tool life assured before the reamer wears down to such an extent that it has to be reground to some other size. The finish obtained is usually

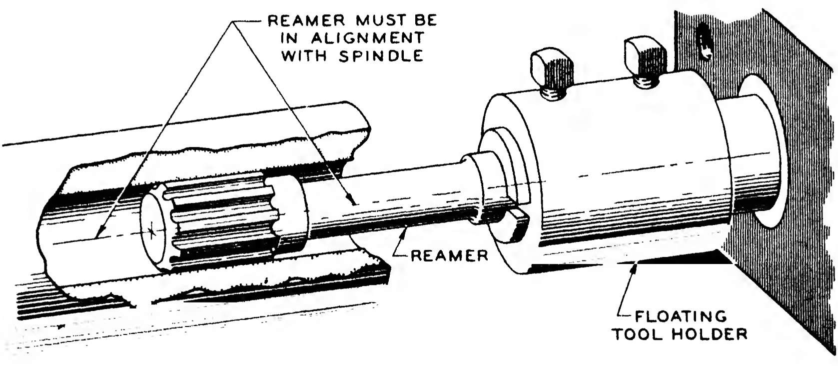


Fig. 16. Illustrating How a Well-Designed, Floating Reamer Works in a Turret Lathe Operation

smooth, providing a good surface for a bearing of the component parts. Reamers tipped with sintered carbides should be used whenever the quantity of work on hand is sufficient to warrant the added expense of the reamer, or whenever hard materials are to be reamed.

Care of Multiple-Edged Tools. All multiple-edged cutting tools, whether they are made of carbon-steel, high-speed steel, of cast carbides, or of sintered carbides, must be handled with meticulous care

to avoid nicks, chips, and bruises in their cutting edges, for these will impair the efficiency of the cutters. This is most important when using sintered carbide tools because nicks and chips are really breaks in the surface of the cutting edge, which, in cutting, will load up and break off more of the cutting edge or perhaps will leave material for the next cutting edge to remove, thus overloading it.

Tools of this type should be handled in wooden boxes which have been compartmented so as to provide a separate inclosure for each tool. Otherwise, their banging about in handling may cause chipping of the tool edges.

Reamers particularly must have exceedingly keen cutting edges in order to finish holes to the desired degree of smoothness and accuracy of size. These edges should be protected during handling. It is only when they are in perfect condition that the best service can be expected of them. Carbide-tipped tool edges are hard and brittle and will chip easily if care is not exercised in handling them. Giving them the proper care at all times will pay dividends in increased performance and in the satisfaction that comes from the knowledge that the work is well done.

Cutting Fluids. The function of cutting fluids is to keep the cutter and the work cool, and to provide some lubrication to the cutting edges, although the value of the latter point is debatable.

Water soluble oils, mineral oil, lard oil, and mixtures of mineral and lard oil have been successfully applied to milling and reaming. The chief requirement of cutting fluids with respect to the carbides is that they not be applied intermittently. Irregular application of a cutting fluid invariably cracks the tip. Another important requirement is that the work and the tool be flooded for efficient cooling. The mere squirting of oil or other coolant on the cutter will harm rather than help it.

The same precautions that apply to a single-point tool where coolants are used, should be observed here. If a cutting fluid cannot be used or supplied in quantity, milling should be done dry. Many machines are not well enough guarded to make possible the application of a cutting fluid because the speed of the cutter will throw the liquid all around the machine. Should this be the case, the cutting must be done dry.

There seems to be the belief, quite prevalent among many engineers and shop men, that coolants are not applicable or are not required in milling. This is far from the truth. The only reason coolants are not applied more extensively to carbide milling is because the machines, cutters, and coolant systems have not been designed to supply and confine the coolant where it is needed.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Define milling.
2. State the three important steps in the development of milling.
3. Define a milling machine.
4. Classify the various types of milling machines.
5. State on what basis milling machines are classified.
6. Classify milling operations.
7. Define a multiple-edged cutting tool.
8. A plain surface 3'' wide and 18'' long is to be machined in a shaper. If there are 35 cutting strokes in one minute, and the feed is 1/32'' per stroke, determine the time required to take one cut on the surface. (Allow 1/8'' for tool overlap at the beginning of the cut.)
9. If, in the preceding question, a milling cutter 6'' in diameter is used and it makes 200 r.p.m. with a feed of 10'' per minute, determine the time necessary to cut when the total distance traveled by the cutter is 18 1/2'' and the whole surface is to be finished in one cut.
10. If the work done with a carbide tipped cutter took 7 minutes as against 23 minutes for a high-speed steel cutter, what is the saving of time in per cent?
11. What are the limiting factors in milling metal with carbide-tipped cutters?
12. Define reaming.
13. How much material should a reamer remove?
14. How can the misalignment of reamers be corrected?
15. What grades of carbides are used for tipping reamers, and on what does the selection depend?
16. What coolants are generally used for milling operations?

REFERENCES

Begeman, M. L. Manufacturing Processes. New York: John Wiley & Sons, Inc., 1942.

Goddard and Goddard Company. General Tool Catalogue D.

National Twist Drill Company. Milling Cutters and Milling. 1926.

War Production Board, Office of Production Research and Development. "Data sheets for May, 1945."

CHAPTER XIII

Cutting Speeds, Feeds, Forces

The Problem Defined. Milling with sintered-carbide-tipped cutters is a high-speed operation.. It is much faster than with cutters made of carbon-steel or even high-speed steel. The reason for this is that carbide-tipped cutters are able to remove a given amount of material in much less time than is common with high-speed-steel cutters. The higher speeds are possible because the sintered carbide, being a hard material, is able to resist abrasion. Since it has great compressive strength also, it is able to resist the forces put on the teeth by the cut.

In milling with sintered carbide, the speed is also usually much higher than in turning even when the material is the same, because the cutter has only one or two teeth in engagement at any time during its revolution. The remaining teeth are turning in the air and are being cooled by this "fanning" action before reengaging the work. In turning operations, the turning tool is in continuous contact with the work. Milling cutters can normally operate at speeds of from two to three times that allowed in turning, for the same tool life.

How the proper speed, feed, and depth of cut can be determined for carbide cutters will be explained in detail in this chapter. Methods of finding the horsepower needed to make a cut under a given set of conditions also will be demonstrated, as well as the procedure for determining the forces acting upon the tool and the length of time required to make a particular cut.

The Cutting Speed. The cutting speed of a milling cutter is the peripheral, lineal speed resulting from rotation and is determined by multiplying the circumference of the cutter (in feet) by the number of revolutions it makes per minute. It is the length of the path a point in the circumference of the cutter makes in one minute's time. The unit of cutting speed is one foot per minute and is usually designated by s.f.p.m. or merely f.p.m., meaning surface feet per minute. The speed can be figured by using the following formula:

$$S = \frac{3.14 \times D \times N}{12}$$

Where

S = the cutting speed in feet per minute
D = the outside diameter of the cutter in inches
N = the number of revolutions per minute made by the cutter
3.14 = the usual constant

As a means of checking the application of this formula, assume it is desired to determine the speed of a 6" milling cutter which is turning at a speed of 400 r.p.m. The solution consists merely of substituting the known values in the formula, resulting in

$$S = \frac{3.14 \times 6 \times 400}{12} = 628 \text{ f.p.m.}$$

In most shop problems, the cutting speed for a given material is recommended. It then becomes necessary to find the number of revolutions per minute the cutter should make to set the machine for the desired speed. This can be found by using a second formula which is merely a simple, algebraic transposition of the first, and in which the symbols have the same meaning as before.

$$N = \frac{S \times 12}{3.14 \times D}$$

The operation of this formula can be checked by assuming a milling cutter is 8" in outside diameter, will cut a given material at 750 f.p.m., and that it is desired to know at what r.p.m. the cutter must operate in order to cut at the prescribed rate. Making substitutions of known values results in

$$N = \frac{750 \times 12}{3.14 \times 8} = 358 \text{ r.p.m.}$$

Carrying this analysis further, it is seen that it is possible to determine the diameter of a cutter which must turn at a specified r.p.m., provided its speed in f.p.m. is known. Again, this formula is evolved from the first and the symbols have the same meaning as before.

$$D = \frac{S \times 12}{3.14 \times N}$$

To check the working of this formula, assume it is desired to know the diameter of a cutter which must turn at 246 r.p.m., 550 f.p.m. The substitutions of known values are made in the formula, resulting in

$$D = \frac{550 \times 12}{3.14 \times 246} = 8.5'' \text{ diameter}$$

The Feed. The feed in milling is the product of the advance or cut made in the work by one tooth during one revolution of the cutter, the number of teeth in the cutter, and the revolutions per minute of the cutter. The feed is usually designated as follows:

f = the feed of work in inches per tooth of the cutter, also known as chip load
F = the feed of work in inches per minute into the cutter

Thus, a feed of 0.008'' per tooth means that the tooth takes a chip out of the work in the direction of feed of 0.008'', while a feed of 60'' per minute would mean that the work advanced under the cutter at a rate of 60'' per minute, or that a surface 60'' in length was machined in one minute.

F, then, is obtained by multiplying the feed per tooth of the cutter, the number of teeth in the cutter, and the cutter r.p.m.

The Depth of Cut. The depth of cut in milling is the thickness of the material being removed. It is usually the distance between the original and the final surface of the material being cut in one pass of the work under the cutter.

The values of cutting speed, feed, and depth of cut vary for different kinds of work and equipment used. A cutting speed should be used which is the highest possible but which is commensurate with economical tool life between grinds. For roughing operations, a somewhat lower speed and coarser feed are used, whereas for finishing, the feed will be finer and the speed higher. The heavy feeds taken in roughing operations must be consistent with the strength and rigidity of the machine, the fixtures, the work itself, and the finish desired.

Determining Proper Feed. In determining the heaviest feed the cutter can carry without breaking or overstraining the machine, without overcrowding the chip space, and without producing an unsatisfactory finish, the cutter should be started at lower feeds. The feed may then be increased until the point is reached where additional increase in chip load would be expected either to stall the machine or dull the teeth too frequently, necessitating frequent regrinding. The feed can then be stabilized for a given job. Experienced operators can usually select a combination of feeds and speeds which will give maximum metal removal at minimum resharpening of tools, and at the same time giving a satisfactory finish. Table I gives the recommended range for milling feeds for steel, using sintered carbide cutters.

The feed of the work into the cutter and the resulting chip load are of great importance because they govern the output or the amount of production of the machine. Thus, assuming that an 8'' carbide-tipped

TABLE I. RECOMMENDED FEEDS FOR MILLING OF STEEL WITH SINTERED CARBIDE CUTTERS

Type of Milling	Feed per Tooth in Inches
Face	0.006 - 0.012
Side or straddle	0.008 - 0.012
Slab	0.008 - 0.012
Slotting	0.006 - 0.010
Saw	0.003 - 0.006

cutter having ten teeth rotates at 350 r.p.m., and that the chip load or feed per tooth is 0.008", the feed in inches per minute can be obtained by multiplying the number of teeth in the cutter by the chip load and the number of revolutions per minute. If T represents the number of teeth in the cutter, N the r.p.m., and f the chip load, then F, the feed per minute, can be determined by this formula:

$$F = fNT$$

Conversely, if the feed per minute, the number of teeth in the cutter, and the r.p.m. are known, the chip load can be found by using this formula:

$$f = \frac{F}{N\,T}$$

If a cutter with an 8" diameter has ten teeth and is rotating at 350 r.p.m., the feed per minute can be figured by substituting the known values in the formula F = fNT, giving

$$F = 0.008 \times 10 \times 350 = 28'' \text{ per minute}$$

Should the feed per tooth be increased to 0.016", the feed per minute, F, would double, and would be 56". This higher feed, and more, is entirely possible with carbide-tipped cutters, provided the power is available at the machine and that the machine is rigid.

Finding the chip load by using the formula

$$f = \frac{F}{N\,T}$$

is a simple matter provided the rate of feed, speed, diameter, and number of teeth are known concerning the milling cutter. As an example, assume it is desired to know the chip load for a 6" cutter having 8 teeth when the cutter makes 250 r.p.m. and the feed is 60" per minute. When these known elements are substituted in the formula, we have

$$f = \frac{60}{250 \times 8} = 0.030'' \text{ per revolution}$$

This feed is easily possible on cast iron and aluminum when carbide-tipped milling cutters are used.

Determining Proper Speed. The correct cutting speed of the carbide-tipped milling cutter depends on a number of variables. These, as have been pointed out before, include

1. The material cut
2. The amount of material to be removed
3. The depth of cut and feed
4. The power available at the spindle of the machine
5. The finish desired
6. The rigidity of the work, the fixtures, and the machine
7. The hardness and toughness of the material cut
8. The cutting fluid used

TABLE II. RECOMMENDED CUTTING SPEEDS FOR SINTERED CARBIDE MILLING CUTTERS

Material to Be Cut	Cutting Speed in f.p.m.	
	Roughing Cuts 3/16 to 1/8 Depth	Finishing Cuts 1/16 or less Depth
Cast iron		
Soft	275-375	325-400
Medium	200-280	275-300
Hard	175-225	225-275
Chilled	50-100	75-175
Malleable cast iron		
Soft	300-375	375-450
Medium	250-300	300-375
Hard	200-250	250-300
Steel, cast		
Soft	200-275	275-400
Medium	175-200	225-350
Hard	100-175	175-250
Steel, rolled		
Soft	350-650	350-650
Medium	250-500	300-500
Hard	150-250	250-300
Brass	350-500	450-700
Bronze	300-450	450-600
Aluminum	1,000-2,000	2,000-5,000
Magnesium	1,000-2,000	2 000-5,000

Recommended speeds for milling cutters for various materials are given in Table II. These speeds are conservative and may be varied to suit operating conditions.

It has been proven through long experimentation that the cutting speed of a milling cutter is a function of the hardness of the material cut. That is, the harder the material, the slower will be the cutting speed for the same tool life. Recommended cutting speeds for milling cutters which are used for cutting carbon and alloy steels heat-treated to various hardnesses, are given in Table III.

The determination of proper cutting speed will be demonstrated in the following example:

A 10" milling cutter having 12 teeth is used for machining hard cast iron. The depth of cut is 3/16". Assume it is desired to determine the cutting speed and r.p.m. of the cutter. The known values are D = 10, T = 12, and d = 3/16. It is learned from Table II that S, the cutting speed, is 175 to 225, or about 200 as an average. The solution consists of substituting these values in the formula used for finding N. This gives

$$N = \frac{12 \times 200}{3.14 \times 10} = 76.5 \text{ r.p.m.}$$

TABLE III. SUGGESTED CUTTING SPEEDS FOR SINTERED-CARBIDE-TIPPED MILLING CUTTERS BASED ON HARDNESS OF WORK MATERIAL

HARDNESS OF WORK			CUTTING SPEED IN F.P.M.	
Brinell	Rockwell C	Shore Scleroscope	Roughing 1/8" to 3/16" Cut	Finishing 1/16" or less Cut
682	65	93	200	250
601	60	83	200	275
545	55	75	250	290
497	50	68	280	350
427	45	62	325	400
370	40	54	350	425
323	35	46	380	475
276	30	42	425	525
249	25	38	455	555
217	20	33	500	610
196	15	30	515	650
180	10	28	560	680
160	5	23	580	700
130	0	—	650	770

Determining Cutting Time. It is frequently necessary to estimate the time required to machine a surface of a certain length. This can be done by substituting the known values in this formula:

$$t = \frac{L}{fNT}$$

Where

t = the time in minutes required to cut the material
L = the length of work to be machined, in inches
f = the feed per tooth in inches
T = the number of teeth in the cutter
N = the r.p.m. of the cutter

The length of table travel under the cutter while the latter is cutting the material is shown in Fig. 1. In examining this figure, it will be

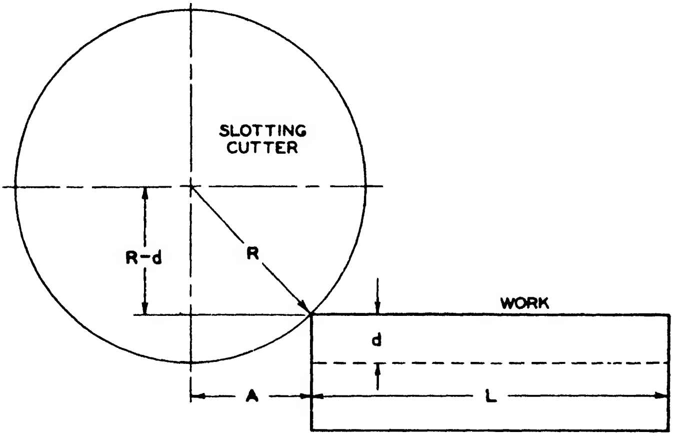


Fig. 1. The Approach Factor in Milling

noted that the travel during the cut includes not only length L, but approach A. Therefore, the total length to be cut is actually L + A. If the formula for determining cutting time is to be strictly correct, this smaller distance must be included, resulting in

$$t = \frac{L + A}{fNT}$$

The approach, A, is determined by another formula which is evident from the geometry of the drawing in Fig. 1.

$$A = \sqrt{R^2 - (R - d)^2}$$

Where

A = the approach of the cutter in inches
R = the radius of the cutter used, in inches
d = the depth of the cut in inches

To check the application of this formula, assume a 6" milling cutter is to be used for cutting a slot in a workpiece. The slot is to be 1" deep. The cutter has 8 teeth and the feed per tooth is 0.008". It is desired to determine the approach, A. The solution is achieved by substituting values in the formula, resulting in

$$A = \sqrt{3^2 - (3 - 1)^2}$$

$$= \sqrt{9 - 4}$$

$$= \sqrt{5} \text{ or } 2.236''$$

This amount should be added to the length of the work to be machined in order to get the accurate length of time required for the job.

For example, if, in the preceding problem the length of the workpiece to be machined is 52", the cutter makes 200 r.p.m., and it is desired to determine the time necessary to make the entire cut, A must be included in the formula. When the known values are substituted in the formula, we have

$$t = \frac{52 + 2.23}{0.008 \times 200 \times 8} = 4.23 \text{ minutes}$$

It is a fully established fact that it is more economical to machine with smaller milling cutters than with larger ones, partly because the approach is less with the smaller cutter, and partly because the smaller cutter, for the same cutting speed, makes a greater number of revolutions per minute than the larger one. This is true even though the larger cutter may have more teeth than the smaller one. To illustrate this point, consider the following problem:

Assume it is desired to cut a slot 1/2" deep in work which is 60" long. The cutting is to be done at 550 f.p.m. with a feed of .008" per tooth. The cutters available are a 4" with 6 teeth and a 6" with 8 teeth. It is necessary to know the time that will be required for milling with each cutter.

As before, the solution consists of substituting the known values for the letters in the proper formula. The unknown elements are the time necessary to take the cut with each cutter, the approach, and the r.p.m.

The approach for each cutter is found by using the formula given in conjunction with Fig. 1. Thus, for the 4" cutter,

$$A = \sqrt{2^2 - (2 - 0.5)^2} = \sqrt{1.75} = 1.32''$$

and for the 6" cutter,

$$A = \sqrt{3^2 - (3 - 0.5)^2} = \sqrt{2.75} = 1.66''$$

The r.p.m. for each cutter is found by making the first transposition of the formula given for determining the cutting speed. Substituting values for the letters in the formula gives

$$N = \frac{550 \times 12}{3.14 \times 4} = 525 \text{ r.p.m.}$$

for the 4" cutter, and

$$N = \frac{550 \times 12}{3.14 \times 6} = 350 \text{ r.p.m.}$$

for the 6" cutter.

These values may now be used in the formula given for determining the time required for milling, giving, for the 4" cutter,

$$t = \frac{60 + 1.32}{0.008 \times 525 \times 6} = 2.43 \text{ minutes}$$

and, for the 6" cutter

$$t = \frac{60 + 1.97}{0.008 \times 350 \times 8} = 2.75 \text{ minutes}$$

A comparison of these two figures shows that the saving of time for the 4" cutter amounts to

$$\frac{2.75 - 2.43}{2.75} \times 100 = 13.1 \text{ per cent}$$

Formula for Face Mill Operations. If the work is done with a face mill, as shown in Fig. 2, the formula given for computing the approach cannot be used. However, a new formula can be developed which is derived from the geometry of the figure under the new conditions.

$$A = R - \sqrt{R^2 - (W/2)^2}$$

The operation of this formula can be checked by assuming it is desired to determine the approach and the time required for one pass of the cutter. The face mill is 10" in diameter and has 12 teeth. It is to be used for facing a piece of work 72" long and 8" wide. The feed per tooth is 1/32", the cutting speed is 450 f.p.m. These known values are substituted for the letters in the formula, resulting in

$$A = 5 - \sqrt{5^2 - (8/2)^2}$$

$$= 5-\sqrt{25-16}$$

$$= 5-\sqrt{9}$$

$$= 5-\ 3 \text{ or } 2$$

To get the revolutions per minute needed for the solution of the problem, the formula

$$N = \frac{S \times 12}{3.14 \times D}$$

is used as before. These values are then combined in the formula given previously for determining the time required to make a cut, resulting in

$$t = \frac{72 + 2}{1/32 \times 12 \times 172}$$

$$= \frac{74}{64.5}$$

$$= 1.15 \text{ minutes}$$

This time, of course, does not include the time required for loading and unloading the work, nor for the measurements of the work to be done.

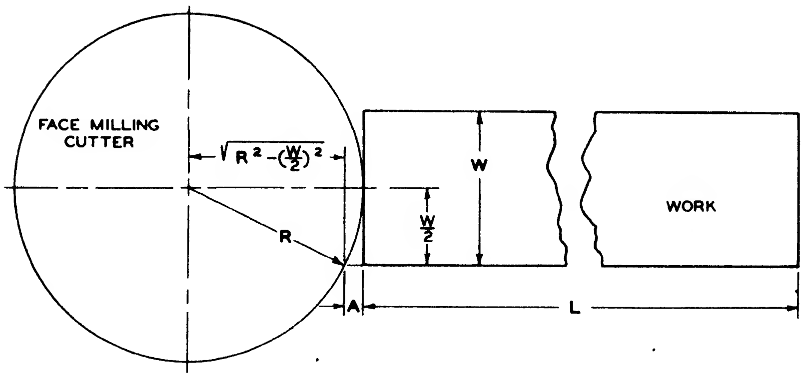


Fig. 2. The Approach Factor in Face Milling Operations

Horsepower Required. The power required for milling is based on the relationship between the horsepower and the rate that material can be removed in cubic inches per minute. Obviously, this is extremely variable and is dependent on many factors such as the cutting speed, the feed, the material cut and its metallurgical and

mechanical properties, the depth of cut, the width, the rake angles of the cutter teeth, the keenness and the smoothness of the cutting edges of the teeth, the type of cutting fluid used, and the condition of the machine and of the work holding fixtures.

A generally accepted value of power for cutting soft steel is 1 hp per .75 to 1.2 cubic inches of metal removed per minute. The number of cubic inches removed per minute can be figured easily from the following formula:

$$C = dfNTw$$

In which

- C = cubic inches of material removed in 1 minute
- d = the depth of cut in inches
- f = chip load, or feed per tooth per revolution, in inches
- N = the r.p.m. of the cutter
- T = the number of teeth in the cutter
- w = the width of the cut in inches

The use of this formula may be checked by assuming a carbide-tipped milling cutter having 8 teeth and revolving at 200 r.p.m. is taking a cut 1'' wide and 1/2'' deep with a feed of .006'' per tooth. It is desired to know how many cubic inches of metal are being removed by the cutter each minute. The known values are substituted for their corresponding letters in the formula, resulting in

$$C = 1/2 \times 0.006 \times 200 \times 8 \times 1 = 4.8 \text{ cubic inches}$$

If 3/4 of a cubic inch of material is to be removed per horsepower per minute, then the horsepower necessary to take such a cut will be approximately 4.8 ÷ 3/4 or 6.4. To this should be added approximately 25 per cent additional power to compensate for the dulling and wear of the cutter, making the total horsepower requirement 8.

This, then, gives rise to the formula which will give the approximate horsepower necessary to cut metal:

$$hp = KC$$

In which

- hp = the horsepower necessary to cut the material
- K = a constant which is dependent on the material cut and the condition of the cutter
- C = the cubic inches of metal removed per minute

If the formula for determining C is substituted for the letter in the formula just given for finding the horsepower necessary to cut metal, we have

$$hp = KdfNTw$$

TABLE IV. APPROXIMATE VALUES OF CONSTANT K

Material Cut and Its Hardness	Constant K
Bakelite	0.2
Brass	0.48
Cast iron	
Soft	0.6
Medium hard	0.8
Hard	1.0
Copper	0.95
Steel	
125 Brinell	1.1
150 Brinell	1.2
175 Brinell	1.3
200 Brinell	1.4
250 Brinell	1.5
300 Brinell	1.6
400 Brinell	1.7
500 Brinell	1.8
600 Brinell	2.0

It should be remembered that K is a constant which is dependent on the material cut and the condition of the cutter. For steel of 500 Brinell hardness, the value of the K would be 1.8. If the horsepower requirements in the problem had been determined by use of the above formula rather than by the rule-of-thumb 1 hp for every .75 to 1.2 cubic inches of steel removed, the necessary power would be

$$\text{hp} = 1.8 \times 4.8 \text{ or } 8.64$$

This is approximately the same figure that was obtained previously.

The values of the constant K vary with the kind of material cut, its hardness, the construction of the cutter, and its condition and sharpness. The approximate values given in Table IV may be used for estimating purposes.

Machine Efficiency. Using the formula hp = KC, the machine operator computes the horsepower necessary to take a cut. This horsepower, however, is the horsepower at the spindle and is less than the horsepower of the motor driving the machine unless one motor is used to drive the spindle and another to drive the feed mechanism. The horsepower necessary to drive the machine and take the cut may be obtained by dividing the formula hp = KC by from 50 to 80 per cent, depending on the condition of the machine. For average conditions, an efficiency of 75% is considered normal.

An example of the use of the power requirement formula can be had

by assuming it is necessary to determine the horsepower required to drive the spindle as well as the total horsepower, when the machine efficiency is 75%. A 6" cutter with 8 teeth is to be used on a cut 4" wide and 1/8" deep. The material to be machined is steel of 250 Brinell hardness. A feed of .008" at 300 r.p.m. is to be used. These values are substituted in the power formula, giving

$$hp = 1.5 \times 1/8 \times 0.008 \times 300 \times 8 \times 4 = 14.4$$

Since the machine is but 75% efficient, this figure must be divided by .75 to arrive at the total horsepower requirements. Thus

$$\frac{14.4}{0.75} = 19.2 \text{ hp}$$

This indicates that the machine should have a 20 hp motor.

The foregoing formulas and constants are approximate, but are adequate for practical purposes, particularly for estimating. Values of constants are approximate because operating conditions vary with the material cut, the type of cutter used, the speed and the feed, the condition of the machine, the depth of cut taken, the kind of coolant used, and many other factors. However, more accurate formulas are available and may be used for horsepower requirement computations where desired. Most of these more accurate formulas were derived from experimental results conducted in laboratories under specific operating conditions. For example, Professor O. W. Boston, in his textbook, Metal Processing, gives the following formula for energy required for milling materials, both up and down, with and without the use of a cutting fluid:

$$E = Cwf^x d^y$$

In which

E = the energy in foot pounds per chip at the tool point
C = a constant for cutter material and cutting fluid
w = the width of the cut in inches
f = the feed in inches, or chip load
d = the depth of the cut in inches
x and y = experimental exponents, varying with the material cut. The same is true of the constant, C.

For milling cast iron, the value of E was found to be as follows:

$$E = Cwf^{.41}d^{.56} \text{ per chip,}$$

where C varies between 569 for milling up, and 643 for milling down.

Having the energy per chip in foot pounds, the horsepower necessary to cut can be obtained by substitution in the following formula:

$$hp = \frac{E\ T\ N}{33{,}000}$$

in which

E = the energy in foot pounds per chip
T = the number of teeth in the cutter
N = the r.p.m. of the cutter
33,000 = the horsepower constant

The application of these more accurate formulas to specific problems is beyond the scope of this book. Those readers interested in such exact and highly complicated methods are referred to the work cited, and to others of a similar nature, listed at the end of this chapter.

The net and/or gross horsepower required for milling materials also can be determined by actual measurements of power consumed during cutting. Divided by the number of cubic inches of material removed per minute, this will give the horsepower required per cubic inch of material removed.

In a research project known as M 580, conducted at the University of Michigan for the American Society of Mechanical Engineers' Manufacturing Engineering Committee, the following formula was obtained as the result of cutting cast-iron bars having a tensile strength of 20,000 p.s.i. and a hardness of 170 Brinell, with a sintered carbide tipped cutter 9'' in diameter having 16 teeth.

$$hp_g = \frac{0.113f^{.71}NTdw}{E}$$

In which

hp_g = the gross horsepower of the machine
f = the feed in inches per tooth
N = the number of revolutions per minute
T = the number of teeth in the cutter
d = the depth of cut in inches
w = the width of the cut in inches
E = the efficiency of the machine (about 70 per cent)
0.113 = a constant

To facilitate the computations of horsepower requirements in the face milling operations involving material and conditions just described, the values of f, raised to the power of 0.71, are given for different feeds in Table V.

The use of this formula can best be appreciated by assuming it is desired to know what horsepower will be necessary for the milling of soft cast iron having a tensile strength of 20,000 p.s.i. and a Brinell

TABLE V. VALUES OF $f^{0.71}$

Feed	$f^{0.71}$
.002	.012
.004	.020
.006	.026
.008	.032
.010	.038
.012	.043
.016	.054
.020	.062
.025	.073
.030	.083

hardness of 170, using a milling cutter 9" in diameter and having 16 teeth. The depth of cut is to be 1/8", the width 4", the feed .004" per tooth, and the speed 200 r.p.m.

The solution consists of making substitutions in the formula, replacing the symbols with the known values, and taking 75 per cent as E, the efficiency of the machine. This results in

$$hp_g = \frac{0.113 \times 0.019 \times 200 \times 16 \times 1/8 \times 4}{.75} = 4.6$$

The application of this formula under slightly more difficult conditions can be appreciated when it is assumed that it is necessary to determine the gross horsepower required when cutting cast iron of about 20,000 p.s.i. tensile strength and having a Brinell hardness of 170. The depth of cut is to be 1/8", the width 4". The cutting speed is to be 350 f.p.m. with 0.008" feed per tooth. The cutter is 9" in diameter and has 14 teeth. The efficiency of the machine is rated at 75 per cent.

The first step in the solution of this problem is to determine the r.p.m. of the cutter. Thus,

$$N = \frac{S \times 12}{3.14 \times D}$$

$$N = \frac{350 \times 12}{3.14 \times 9}$$

$$N = 148 \text{ r.p.m.}$$

The known values can now be substituted in the formula, resulting in

$$hp_g = \frac{0.113 \times 0.032 \times 148 \times 14 \times 1/8 \times 4}{0.75} = 4.9$$

TABLE VI. SOME VALUES OF $f^{0.77}$

f	$f^{0.77}$
.002	.0084
.004	.0142
.006	.0196
.008	.024
.010	.029
.015	.040
.020	.049
.025	.058
.030	.067

For the face milling of Meehanite cast iron of 40,000 p.s.i. tensile strength and 190 Brinell hardness, the following formula is given for horsepower requirements at the cutter:

$$hp_c = 0.18 \times f^{0.77}NTdw$$

In this formula, hp_c is the horsepower at the cutter, the other symbols being the same as previously. Some values for $f^{.77}$ are given in Table VI.

The function of this formula can be checked by assuming that it is desired to know the horsepower at the cutter when the following conditions and job specifications exist: the material to be machined is Meehanite of 40,000 p.s.i. tensile strength, 190 Brinell hardness; the cut is 4'' wide, 1/8'' deep the feed is to be 0.008'' per tooth; the cutter has 16 teeth and is rotating at 175 r.p.m. After the substitutions are made in the formula, we have

$$hp_c = 0.18 \times 0.125 \times 0.0196 \times 175 \times 16 \times 4 = 4.93$$

It is possible to derive a second formula from that used to determine the horsepower at the cutter, and thereby find the value of T (the number of cutter teeth) when it is not given. Thus,

$$T = \frac{hp_g \times E}{0.113 \times f^{0.71}Ndw}$$

If, for example, the job specifications call for a cut 1/4'' deep and 6'' wide to be made with a carbide tipped cutter at 100 r.p.m. in Meehanite of 190 Brinell hardness, using a machine of 15 hp which is 75 per cent efficient, it will be possible to determine the correct number of cutter teeth necessary to complete the job without overtaxing the machine.

Substituting known values for letters in the formula gives

$$T = \frac{0.75 \times 15}{0.113 \times 0.058 \times 100 \times 0.025 \times 6} = 12$$

It should be noted that an increase of speed will bring about an increase in the horsepower necessary to make the cut. The same is true, of course, with increase of feed, depth of cut, and width of cut. The demand for horsepower can be reduced, on the other hand, by making the cutter with fewer teeth.

Rake Angles and Power. Much has been written on the advantages of milling with cutters having a negative rake angle and on the power consumed during such milling. Negative rake angle cutters, particularly when used in milling hard, strong, and tough materials, have proved superior to cutters having positive radial rakes so far as the life of the tool is concerned. However, they unquestionably consume more power than cutters with positive radial rake angles or even cutters with zero radial rake.

If the power requirements for a cutter with zero radial rake are 100, the power required by the cutter with 10° positive radial rake may be from 12 to 20 per cent less. Again, for a cutter with 10° negative radial rake, the power requirements may be from 12 to as high as 20 per cent more than those for the cutter with zero radial rake.

Therefore, strictly from the standpoint of power requirements, the positive radial rake angle cutter has an advantage over the one with the negative radial rake. However, the negative rake cutter may leave a better finished work surface and unquestionably will last longer under severe operating conditions.

Tool force, that is, the force set up on the cutter in making the cut, is less in a cutter with a positive radial rake than in one with a negative rake angle. If the force on the cutter tooth is taken at 100 for zero radial rake, the cutter with 10° positive radial rake will cut with less force by 12 to 20 per cent, while a cutter with 10° negative radial rake will develop from 12 to 20 per cent more force during cutting.

Table VII gives approximate percentages of forces on the cutter teeth, together with the horsepower necessary to cut for cutters with different radial rake angles.

Forces Acting in Cutting. The cutting force is the pressure that compresses the material ahead of the cutting edge of the tool or cutter and shears it off in the form of a chip. This force is dependent on the cutting resistance of the material being machined and on the frictional resistance of the tooth face to the sliding of the chip.

Many studies have been made of the forces acting on the milling cutter tooth and much has been published on the values of the cutting forces, based upon these experimental results. However, to date there have been no simple formulas developed for the exact computation of

TABLE VII. PERCENTAGES OF TOOL FORCES AND HORSEPOWER FOR POSITIVE AND NEGATIVE RADIAL RAKE CUTTERS

Radial Rake Angle	Tool Force in Per Cent	Horsepower in Per Cent
+ 10	88	88
+ 5	94	94
0	100	100
− 5	108	108
− 10	114	114
− 15	118	118

the forces acting upon the milling cutter teeth, based on such known values as the depth of cut, the feed, the radial and the axial rakes of the teeth, and the strength of the material being cut. Commonly, the values of the cutting forces are found from experimental results. These values are then plotted on a graph. The forces can then be taken from these curves and the formulas developed. These formulas can be used for purposes of computation when cutting similar materials under similar operating conditions.

The force in the direction of the cut can be measured, however, with an instrument known as a dynamometer. This method calls for a number of tests under varying conditions of speed, feed, depth and width of cut, kind of material, and other factors. A simpler way is to measure the electrical energy input required for the operation of the cutter and of the machine. From these measurements, computations can then be made of the cutting force on the cutter teeth. However, this method involves the consideration of both the mechanical and electrical efficiency of the machine and the motor. The values obtained, though not "scientific" are sufficiently correct for shop conditions. Since proper electrical energy measuring instruments are not commonly available in the average shop, even this method does not lend itself readily to practical use,

Computation of the cutting force is necessary first, so that the cutter may be designed strongly enough to make the cut, and second, so that the tool engineer will know what forces are acting on the work and the work holding fixtures.

The horsepower required to make a particular cut has been discussed and computed in the preceding paragraphs. Horsepower, as has been seen earlier, is the energy required to move 33,000 pounds one foot in one minute. Work, in a mechanical sense, is a force multiplied by the distance through which the force acts, measured in foot pounds. Work in foot pounds, then, takes no account of the time element. From these facts, it is possible to obtain the cutting force, using the simple horsepower formula

$$hp = \frac{P \times S}{33{,}000}$$

in which

P = the cutting force in pounds
S = the speed at which cutting is done in feet per minute
33,000 = the horsepower constant

From this formula, force P can be obtained by solving for it algebraically. Thus

$$P = \frac{hp \times 33{,}000}{S}$$

The force computed will be found entirely satisfactory for average shop conditions.

In order to check the application of these two formulas, assume an 8'' face milling cutter is cutting steel of 150 Brinell hardness and revolving 310 times per minute while taking a cut with .005'' feed at a depth of 1/8''. The cutter has 8 teeth. The width of the cut is 4''. It is desired to know what horsepower will be necessary to make the cut and what the cutting force will be.

The known values are substituted in the formula given for determining horsepower, resulting in

$$hp = 1.2 \times 1/8 \times 0.005 \times 310 \times 8 \times 4 = 7.4$$

In order to find what P or the cutting force is, it will be necessary first to find the cutting speed. This is done by substituting the known values for the letters in the formula given in the first part of this chapter. Thus,

$$S = \frac{3.14 \times 8 \times 310}{12} = 649 \text{ f.p.m.}$$

With the speed known, it is possible to solve for P. Thus

$$P = \frac{7.4 \times 33{,}000}{649} = 377 \text{ pounds}$$

This is an average cutting force.

In milling operations, the cutting forces in the direction of the cut are made up of the forces necessary to shear the material approximately along the plane indicated by the line X-X in Fig. 3, and from the force necessary to overcome the frictional resistance of the tool against the chip. The shearing force along the line X-X is approximately equal to the distance O-P in inches, multiplied by the width of the cut in inches, and by the shearing strength of the material being cut.

The frictional force, acting in the direction Y-Y, is the force chiefly responsible for erosion of the tool face, an action commonly known as cratering. Cratering is particularly noticeable when a tough, strong material is being cut. The cutting force in the direction of the cut is shown by line Z-Z and is the result of the two forces just described. The cutting force is known to increase slightly with any increase of cutting speed. The horsepower, on the other hand, increases directly with the increase of speed. That is, when the speed is doubled, the horsepower necessary to cut the material, other conditions remaining the same, is also doubled,

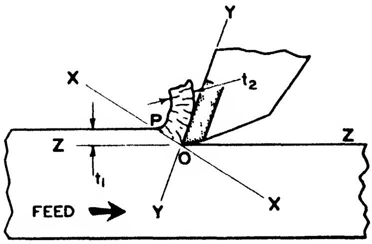


Fig. 3. The Cutting Action of the Positive Radial Rake Cutter Tooth

Forces on the Negative Rake Tooth. For the cutter with a negative rake, the forces on the tooth are similar to those for the cutter having teeth set with a positive rake. That is, the forces along lines X-X and Y-Y in (A) of Fig. 4 are similar to those shown in Fig. 3. The resulting cutting force in the direction of the cut is illustrated by the line Z-Z.

It will be seen in (A) of Fig. 4 that the chip thickness t_2 is greater than the depth of cut, indicated by t_1, revealing a deformation in the chip. It is evident, then, that if the chip is thicker, it will require a

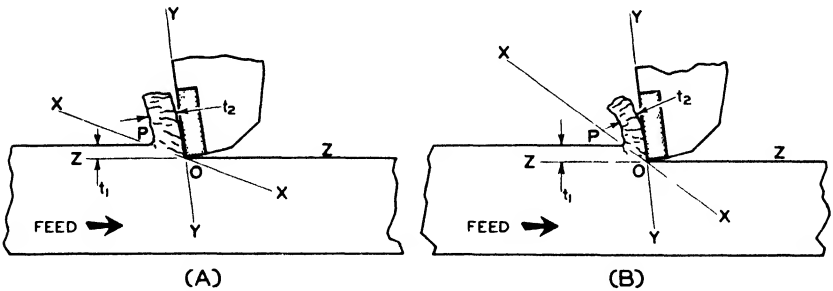


Fig. 4. The Cutting Action of the Negative Rake Cutter Tooth, Showing How the Chip Thins at High Speed

greater cutting force. However, it has been found that with an increase in the cutting speed the chip thickness becomes less than that shown in (A) of Fig. 4, and more like that shown in (B). This calls for less force to do the cutting. This thinning of the chip is one reason why cutters with negative rake angles are operated at higher speeds.

Careful observations of the negative radial rake tooth cutter at these higher speeds point to a very interesting explanation of the reason for the reduced frictional force along the tooth face. This reduction in

frictional force occurs when the thinner chip is formed because of the high cutting speeds. The reasoning is that, at higher speeds, the temperature of the metal at the point of cutting is raised to a degree high enough to soften it, making it less resistant to the shearing force. The validity of this reasoning will be determined with future experimentation and investigation.

Interesting results also are obtained when the cutting forces, acting on the cutter teeth, are plotted against the speed for both positive and negative radial rake angles. Such a graph will show that, for the positive rake cutter, the cutting force increases with the speed, while there is a less gradual increase of the cutting force with the increase of the speed for the negative rake cutter. This is one of the reasons why milling with negative radial rake angle cutters is popular and practicable at speeds hitherto believed impossible. The other reason for using negative rake angle cutters, of course, is that the tooth starts cutting back of its vulnerable face, giving it longer life.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. Compute the cutting speed of a milling cutter 6'' in diameter making 350 r.p.m.

2. If the cutting speed of a 5'' milling cutter is to be 650 f.p.m., what should its r.p.m. be?

3. A 5'' milling cutter tipped with sintered carbide, having 6 teeth, revolves 580 times per minute. If the feed per tooth or the chip load is .006'', what is the feed of work into the cutter in inches per minute?

4. An 8'' milling cutter having 10 teeth and running at 300 r.p.m. cuts a distance of 60'' in one minute. Find the feed per tooth or the chip load.

5. What is the recommended speed for a cutter milling steel of 325 Brinell hardness?

6. A 6'' slot milling cutter having 8 teeth takes a cut 1'' deep with a feed of .004'' per tooth, at 450 f.p.m. The length of the cut is 30''. Determine the following: a) the r.p.m. of the cutter, b) the approach of the cutter, c) the time necessary to take the cut.

7. A 16'' face mill making 125 r.p.m. is cutting cast iron, taking a depth of cut of 1/4''. The feed is 48'' per minute, and the width of cut 12''. Determine: a) the cutting speed in f.p.m., b) the approach in inches, and c) the time necessary for a cut 72'' in length.

8. A cast-iron base for a machine 18'' wide and 6' long is to be faced over its entire length. The recommended cutting speed is 480 f.p.m., and the feed is 1/32'' per tooth. Two milling cutters are available. No. 1 is 24'' in diameter and has 26 teeth. No. 2 is 30'' in diameter and has 26 teeth. Determine: a) the r.p.m. for each cutter, b) the

approach for each cutter, c) the time necessary to take the cut with each cutter, d) the per cent of time saved by one cutter over the other.

9. A 10'' face mill having 12 teeth, tipped with sintered carbide, takes a cut 1/4'' deep and 8'' wide at a feed of .008'' per tooth. The cutter is making 200 r.p.m. Determine: a) the feed in inches per minute, b) the cubic inches of material removed per minute.

10. If the cutter in question 9 removes 1.6 cubic inches of material per horsepower per minute, determine the horsepower necessary for cutting.

11. A 6'' face milling cutter having 8 teeth takes a cut 1/8'' deep and 3'' wide, at a feed of .008'' per tooth, at 300 r.p.m. Determine: a) the cubic inches of material removed per minute, b) the horsepower necessary to cut if the hardness of the workpiece is 200 Brinell.

12. The horsepower for a cutter having zero radial rake was found to be 16. What horsepower would be necessary to do the cutting under the same conditions if the cutter were changed to a 10° negative radial angle?

13. A milling cutter having 12 teeth and making 300 r.p.m. cuts through 60'' of metal. What is the feed per tooth?

14. If the cutter in question 13 takes a cut 1/4'' deep and 6'' wide, how many cubic inches of metal are removed per minute?

REFERENCES

Armitage, J. B., and Schmidt, A. O. "An Investigation of Radial Rake Angles in Face Milling." Transactions, A.S.M.E., Vol. 66, No. 8, Nov., 1944.

——— "Radial Rake Angles in Face Milling," Mechanical Engineering. Vol. 67, No. 6, June, 1945.

Boston, O. W. Metal Processing. New York: John Wiley & Sons, Inc., 1941.

Ernst, Hans, and Merchant, M. E. "Chip Formation, Thickness, and Finish," Cincinnati Milling and Grinding Machines, Inc.

Firth-Stirling Steel Company. "Hyper Milling," Engineering Bulletin No. FE-106.

General Electric Co. "The Motorule," Circular GES-3085.

Martelloti, M. "An Analysis of the Milling Process, Part II—Down Milling," A.S.M.E., Vol. 67, No. 4, May, 1945.

Merchant, M. E. and Zlotin, Norman. "Nomographs for Analysis of Metal Cutting Processes," Mechanical Engineering, Vol. 67, No. 11, November, 1945.

Schmidt, A. O. "Data on Production Runs with Carbide Milling Cutters," reprint of paper for fall meeting of the Cincinnati Section, A.S.M.E., 1945.

War Production Board, Office of Production Research and Development. "Data Sheets for August, 1945."

CHAPTER XIV

Multiple-Edge Tool Design

The Scope of High-Speed Milling. High-speed carbide milling is being done all over the world on an ever-increasing scale. Users of carbide-tipped cutters enjoy higher production rates at lower cost per unit, longer life of the cutter between grinds, better surface finish of the work, and production that is to much closer tolerances than is possible with high-speed-steel cutters. However, these benefits require more exacting conditions in the design of milling cutters, the cutter holding devices, work holding fixtures, and the machine itself.

Although wide use has been made of sintered carbide in milling operations for many years, generally little is known about the laws governing the cutting action of the milling cutter as they relate to the kind and condition of the material to be cut. Similarly, little is known concerning the relation of cutting angles to cutter performance. However, there are engineers who have made extensive studies and researches in milling with carbide-tipped cutters. While these men are familiar with the laws governing such cutting action, the knowledge they have gained has not been widely circulated. Most of the people in the field have little knowledge of what has been happening in the laboratories. This chapter will attempt to supply some of this information.

It is also true that in tool engineering, at least so far as it applies to milling, there is probably more theorizing done than in most fields. That this should be true is easy to understand, since less is known about the cutting of metals by milling than any other metal cutting process. One reason for this state of affairs is that milling consists of a series of interrupted cuts which, in itself, introduces a factor entirely different from, and in addition to, those encountered in drilling, turning, facing, boring, shaping, or planing. Yet, considerable progress has been accomplished in milling even with the limited amount of exact knowledge that has been available.

Factors in Designing. It is generally known that many factors affect the efficiency of a milling cutter, "efficiency" being the amount of material removed per unit of energy used. The most important of these factors have been listed in the following and must be considered carefully when the cutter is designed.

1. The material being milled: its hardness, toughness, and structure.

2. The milling cutter design: the number of teeth, the shape of the teeth, and the rake and relief angles.

3. The nature of the cut—whether it is on the face, the periphery of a cutter, or in a slot.

4. The direction of rotation of the cutter in relation to the table feed.

5. The cut itself: the feed, the width, and the depth.

6. The shape of the chip cut, governed by the chip per tooth, the depth of cut, and the direction of cut.

7. The cutting fluid used, or whether the cutting is being done dry.

8. The power and efficiency of the machine and the driving mechanism.

9. Condition of the machine, the cutter, and of the cutter holding devices.

10. Condition of the work holding devices.

The one outstanding characteristic of milling with sintered carbide tipped cutters is that material can be removed much faster than when cutters of any other material are used. Because more material is removed in a given unit of time, more work is done. However, this creates a greater demand for power at the spindle of the machine. This increased power requirement can be handled by equipping the machine with a larger capacity motor, or designing a milling cutter that will make full use of the power available.

The Number of Teeth. The carbide method of milling is different from high-speed-steel milling in that it removes the material much faster in a given time. Obviously, this faster way of doing work consumes more power. If this power is not available, the machine may stall, resulting in a damaged cutter. In all instances of machining operations by milling, it is essential that the speed of the cutter, the number of teeth in the cutter, and the chip load for a given depth and width of cut be so selected that the demand for power does not materially exceed the rated horsepower of the motor that drives the machine, and does not place undue stress on the power transmission.

The preceding chapters of this book have dealt with the theories and practices underlying cutting speed, chip load, the feed rate in inches per minute, the horsepower necessary to make the cut, the cubic inches of material removed, and the force acting on the cutter teeth. It would be well at this time to review some of the points developed previously in preparation for the study that follows.

When planning the design of a carbide cutter, the engineer should bear in mind that it is desirable to have the size of a cutter for a given job such, that for the given speed at which the cutting is to be done, the number of revolutions per minute will be more than 100. This will permit an increase of speed and an increase in power demand without too great a magnification of the torque. Higher speed causes the cutter to perform more smoothly, a desirable situation from the point of view of wear and tear on the machine. Smoothness of operation also can be achieved by the application of a flywheel, located on the spindle of the cutter or on the arbor holding the cutter. The flywheel should be placed

as close to the cutter as possible without interfering with freedom of cutting. The flywheel stores energy during the time when there is less demand for power in the cycle of cutting, releasing it during that part of the cycle when there is a greater demand. This serves to smooth out the effect of the intermittent cut. For this reason, the use of a flywheel permits a more efficient employment of power by the machine and results in better cutting action by the cutter.

Computations for Teeth. With carbide milling, then, speeds are higher and chip loads are greater than for cutters made of high-speed steel. This calls for greater power at the spindle, and, in order that the machine not be overtaxed, requires that certain factors be controlled. These may be the depth and width of the cut, but if they are also fixed by the requirements of the job, the only other factor that can be changed is the number of teeth in the cutter, although the speed of the cutter can occasionally be varied also. It should be borne in mind that carbide cutters must operate at speeds roughly three to four times those commonly used for high-speed-steel cutters if their full efficiency and economy is to be realized. The feeds for carbide cutters should be from .002" to .012".

As a means of understanding the problems involved in milling with carbide, let us suppose that certain set conditions exist with regard to a particular situation. The solution can then be worked out in a step-by-step manner.

Thus, for example, assume that a milling machine available for a given job is powered with a 15 hp motor, and that the job consists of milling steel brackets 4" wide and 14" long. A 1/8" thickness of material is to be removed and the hardness of the brackets is approximately 150 Brinell.

In approaching this problem, reference should be made first to tables which have been given previously. These tables contain recommended speeds which are based on the hardness of the material to be cut. From them, it is found that the cutting speed for material of 150 Brinell hardness should be from 580 to 700 f.p.m. (from Table III, Chapter XIII). For a conservative speed, 600 f.p.m. is selected and the rest of the computations will be based on that figure. The cutter diameter was not specified in the problem but it should be wider than the width of the work to be surfaced. Thus, a cutter 5" or 6" in diameter will do the work in one cut. Assuming the choice of a 6" diameter cutter to be desirable, the r.p.m. can be found which will give the cutter the desired speed of 600 f.p.m. This results in

$$N = \frac{S \times 12}{3.14 \times D} = \frac{600 \times 12}{3.14 \times 6} = 382$$

This figure is well above the 100 r.p.m. minimum which was recommended in preceding paragraphs.

Since the horsepower of the machine is known, the maximum number of teeth can be calculated by using the following formula:

$$T = \frac{hp}{KdfNw}$$

In which

T = the number of teeth in the cutter
hp = the horsepower necessary to do the machining
K = a constant
d = the depth of cut in inches
f = the feed per tooth in inches
N = the r.p.m. of the cutter
w = the width of the cut in inches

The commonly recommended feeds per tooth for milling steel with carbide cutters are from a minimum of .003'' to a maximum of .012''. Taking .006'' as an average value, and 1.4 for factor K from Table IV in Chapter XIII, these and the other known values are substituted in the formula just given, resulting in

$$T = \frac{15}{1.4 \times .125 \times .006 \times 382 \times 4} = 9.35$$

Since milling cutters are usually made or supplied with an even number of teeth, a cutter could be selected having either 8 or 10 teeth. However, it is desirable for any cutter to be able to perform a variety of jobs under varying conditions of cutting speeds and feeds. For this reason it would be advantageous to learn how many teeth a cutter should have, all other conditions remaining constant, for a feed of .008'' per tooth. This value is substituted for the feed previously used, resulting in

$$T = \frac{15}{1.4 \times .125 \times .008 \times 382 \times 4} = 7$$

From this, it can be seen that a cutter with 6 or 8 teeth would be satisfactory.

These computations indicate that for a given horsepower available at the machine, a certain maximum performance relative to the depth of cut, the feed per tooth, revolutions per minute, and the width of the cut, may be expected of a cutter having a definite number of teeth. If any of the factors such as the constant K, the depth d, the feed per tooth f, the r.p.m. N, or the width of cut W, is changed, the number of teeth in the milling cutter will have to be changed. If any of these factors is increased, the number of teeth will have to be decreased in order to prevent the machine from being overloaded.

The relationship between these critical aspects of milling procedures can best be illustrated by assuming it is desired to know what r.p.m. the cutter should make in order that the 7.5 horsepower capacity of the machine not be exceeded, when steel castings of 140 Brinell hardness

are to be face milled with an 8'' cutter having 10 teeth. The depth of cut is to be 1/4'', the feed per tooth .005'', and the width of the cut 6''.

In order to arrive at a solution of this problem, a transposition must be made in the formula that was given for determining the number of teeth in a milling cutter. Thus

$$N = \frac{hp}{KdfTw}$$

The known values are substituted in this formula, resulting in

$$N = \frac{7.5}{1.4 \times .250 \times .005 \times 10 \times 6} = 72 \text{ r.p.m.}$$

Obviously, this speed is inadequate since it falls below the 100 mark which was described previously as being the minimum speed at which carbide cutters could efficiently operate. The r.p.m. could be increased without altering the horsepower requirements either by using a cutter of the same diameter but with fewer teeth, or by decreasing the feed, which already is low.

If the former alternative were selected and a cutter with 6 teeth substituted for the 10-tooth cutter, the formula, with substituted values, would be

$$N = \frac{7.5}{1.4 \times .250 \times .005 \times 6 \times 6} = 120 \text{ r.p.m.}$$

The cutting speed now must be checked to determine whether it is too high or too low. Substituting for unknown values in the formula results in

$$S = \frac{3.14 \times 8 \times 120}{12} = 251.2$$

This, also, is too low a figure for the machining of soft steel.

To increase the cutting speed for this size cutter, an increase in r.p.m. should certainly be made. However, this increase must be consistent with the power available at the machine. An examination of the formula given for determining r.p.m. shows that three factors can be changed to achieve greater speed. These are the depth of cut, the feed, and the number of teeth.

Changing the depth of cut would necessitate taking two cuts to finish the work, a procedure which may be used in an emergency but which is not often economical. The feed per tooth may be dropped to .004'' or even .003'', but this is undesirable because with the heavier feed, the chip is removed with less expenditure of energy per cubic inch of material machined, and, in addition, the thinner chip has a tendency to wear out the teeth faster. The number of teeth can be further reduced to 4 or even 2. If the feed were reduced to .004'' and the number of teeth to

4, and these values were substituted in the formula, the result would be

$$N = \frac{7.5}{1.4 \times .250 \times .004 \times 4 \times 6} = 223 \text{ r.p.m.}$$

and the cutting speed would be

$$S = \frac{3.14 \times 8 \times 223}{12} = 466 \text{ f.p.m.}$$

This may well be the best speed for steel castings, for, although soft, they frequently contain sand inclusions which would have a tendency to wear the teeth excessively if the cutter were operated at high speeds.

Another possibility is to take advantage of the overload capacity of the motor. This ranges from 25 to 50 per cent, sometimes as high as 100 per cent, and permits the use of greater feed, speed, depth of cut, and even width of cut. However, this practice is not recommended and should be used only as a last resort, since continued overloading of the machine will cause it to wear out prematurely.

For quick computation of the horsepower required to cut, the constant, k, may be assumed to be 1. This holds true for milling soft steel with a well-designed cutter. However, as the cutter dulls, there is a greater demand for power. For this reason the constant, as given in Table IV in Chapter XIII, represents a more conservative estimate and should be used for computations for power consumed, number of teeth in the cutter, feed, and depth of cut that can be taken with the cutter when other known conditions remain constant. The solution of the depth of cut, the feed, and the number of cuts can be had from transpositions of the formula given for determining the maximum number of teeth for a cutter. These are

$$d = \frac{hp}{KfNTw}$$

$$f = \frac{hp}{KdNTw}$$

$$w = \frac{hp}{KdNTf}$$

These formulas, together with those previously given, constitute a set commonly used in all computations as they relate to milling and milling cutters in general. The symbols have the same meanings as given before.

As a means of increasing proficiency in the use of these formulas, illustrative examples have been worked out in the following problems:

Assume it is desired to find what depth of cut can be taken with a milling cutter 6" in diameter, having 8 teeth, operating at 480 f.p.m., chip load of .005", width of cut of 3", when cutting grey cast iron in a machine powered with a 5 hp motor. The known values are substituted

first in the formula given for determining the r.p.m., or N. The value for K is obtained from Chapter XIII and is considered to be .8. Thus,

$$N = \frac{S \times 12}{3.14 \times D} = \frac{480 \times 12}{3.14 \times 6} = 305 \text{ r.p.m.}$$

This value and the other known values are substituted in the formula used for finding the depth of cut, resulting in

$$d = \frac{5}{.8 \times .005 \times 305 \times 8 \times 3} = .171''$$

This is nearly 3/16''.

In milling hard cast iron in a machine powered with a 5 hp motor, it is desired to learn what feed per tooth may be taken without overloading the machine when a 6'' face mill having 8 teeth is used at a cutting speed of 240 f.p.m. on a cut 3'' wide and 1/8'' deep. Again, in order to arrive at a solution, it is necessary first to determine at what r.p.m. the cutter is operating. Thus, the known values are substituted in the formula, giving

$$N = \frac{240 \times 12}{3.14 \times 6} = 153 \text{ r.p.m.}$$

This value is substituted in the formula given earlier for finding the feed, resulting in

$$f = \frac{5}{1.2 \times .125 \times 153 \times 8 \times 3} = .009''$$

In milling brass with a machine powered with a 5 hp motor, and using a 6'' face mill having 8 teeth, it is desired to know what width of cut can be taken without overloading the machine when the depth of cut is 1/8'', the speed at 1,000 f.p.m., and a feed of .006'' per tooth. Since the r.p.m. is unknown and must be had in order to solve the problem, the first step is to solve for N.

$$N = \frac{1{,}000 \times 12}{3.14 \times 6} = 637 \text{ r.p.m.}$$

This value can now be substituted in the formula given for finding w.

$$w = \frac{5}{0.48 \times .125 \times 8 \times 637 \times .006} = 2.72''$$

It should be noted that if the speed were reduced to half that given in the example, say to 500, a reasonable speed, the r.p.m. of the cutter would be 318, and the width of the cut could be increased to 545'' without overloading the machine.

When given conditions remain constant, the formula used for deter-

mining N can be included in the formula used for finding the number of teeth in the cutter, as follows:

$$T = \frac{hp}{Kdfw \times \left(\frac{S \times 12}{3.14 \times D}\right)}$$

All symbols have the same meanings as described previously.

This formula may be written in still another form, giving

$$\frac{T}{D} = \frac{3.14\ hp}{12\ KdfSw}$$

For a given condition where the number of teeth in the cutter and the cutter diameter have been established, T and D can be varied provided they are always kept equal to the values in the right-hand side of the formula. $\frac{T}{D}$ then, represents a constant, either component of which may be varied at will so long as the quotient is equal to that of the other fixed conditions. Furthermore, $T = D \times$ the constant, and $D = \frac{T}{\text{constant}}$.

The relationship between the constant $\frac{T}{D}$ and the other fixed conditions of a problem can best be shown by an example. Assume it is desired to learn what may be the diameters and the number of teeth in the cutters to be used on a machine powered with a 10 hp motor. The material to be machined is soft steel. The depth of cut is to be 3/32", the width 3", the constant K, is 1, the speed is 600 f.p.m., and the feeds are .006", .008", .010", and .012".

Solving first for the feed of .006", the known values are substituted in the right side of the formula, giving

$$\frac{T}{D} = \text{Constant} = \frac{10 \times 3.14}{1 \times .0937 \times 600 \times .006 \times 3 \times 12} = 2.59$$

It should be noted that the fraction, 3/32, was changed to the decimal, .0937. In the same manner, the constants for feeds of .008", .010", and .012" are found. These equal respectively, 1.94, 1.55, and 1.29. If T is assumed, D can be found by dividing T by the constant. Thus, for a cutter with 10 teeth, the diameter for the 2.59" constant would be $\frac{10}{2.59} = 3.78$, or nearly 3 7/8". Similar computations can now be made for the number of teeth and the diameter of the cutter, or one can be assumed and the other computed. This can be done for every constant, then arranged in tabular form for future reference. Such a tabulation is given for convenience in Table I.

Obviously, some of the values from Table I may be found unsatisfactory for use, since as the diameter increases, the number of revolutions per minute decreases. The r.p.m. preferably should not drop

TABLE I. VALUES OF T AND D FOR CERTAIN OPERATING CONDITIONS

T	Values of D in Inches			
	f = 0.006 C = 2.59	f = 0.008 C = 1.94	f = 0.010 C = 1.55	f = 0.012 C = 1.29
10	3 7/8	5 1/8	6 1/2	7 3/4
12	4 5/8	6 1/8	7 3/4	9 3/8
14	5 1/2	7 1/8	8 7/8	10 1/8
16	6 1/8	8 1/4	10 1/4	12 1/2
18	7	9 1/2	11 5/8	14
20	7 3/4	10 1/4	13	15 1/2

below 100. Thus, for a cutter 12 1/2" in diameter, the r.p.m. should be equal to $\frac{600 \times 12}{3.14 \times 12.5}$ or 183. For a cutter 15 1/2" in diameter, the r.p.m. would be 148. Both these values are above the minimum set forth and can be used if necessary. However, the smaller diameter cutter would be preferable because of the greater number of revolutions per minute it makes. As was pointed out in Chapter XIII, the smaller cutter takes less time in doing the work.

Cutter Size and Shop Practice. The number of teeth in the milling cutter as computed by the use of the first formula given in this chapter and illustrated in the preceding pages is practical, but applies to particular applications. For average shop work, that is, when the cutter is to be used for general work on different steels and other materials as is usually the case with most smaller organizations, the size of the cutter, when computed as explained, may not answer the purpose. However, it can be used by taking a smaller or larger chip load, varying the depth of the cut, and regulating the speed to meet the condition imposed by the operation. This point will be discussed later in the chapter.

The number of teeth in the carbide cutter used for milling steel is generally fewer than in the traditional high-speed-steel cutter. Experience indicates that the number of teeth in a cutter which is suitable for a wide range of milling applications such as those found in the general shop, may be determined by an empirical rule or a sort of "rule-of-thumb" method. This rule is that the number of teeth in a carbide-tipped milling cutter should be equal to the diameter of the cutter in inches, plus 2. Stating this as a mathematical formula gives

$$T = D + 2$$

This means that a cutter 4" in diameter would have 6 teeth, a 6" cutter would have 8 teeth, a 7" cutter 9 teeth, and so on. On the other hand, some tool engineers prefer the number of teeth in the carbide cutter to be equal to the number of inches of diameter, that is, T = D. Such cut-

TABLE II. NUMBER OF TEETH IN HIGH-SPEED STEEL AND CARBIDE-TIPPED CUTTERS

Diameter of Cutter in Inches	Teeth in High-Speed-Steel Cutters	Teeth in Carbide-Tipped-Cutters
3	8	4 to 6
4	10	6
5	12	8
6	12	8
7	12	10
8	14	10
10	16	12

ters have performed equally well when other conditions are satisfactory.

Table II gives the relative number of teeth in high-speed milling cutters used for heavy work, and in carbide-tipped cutters used for roughing and finishing applications in an average production shop.

The number of teeth in engagement with the work is sometimes a vital factor in the milling operation. Where there is only one tooth in contact with the work, the operation becomes jumpy and the pounding of the teeth as they engage the work brings about intermittent stresses on the machine parts and variable loads on machine bearings. To gain smoothness of operation, many engineers prefer a finer pitch tooth. By this is meant a greater number of teeth for a given diameter of the cutter. Also recommended are lighter chip loads, more conservative speeds in milling applications, and a lesser depth of cut. These factors reduce the power requirements and in most cases increase tool life between grinds. Also increased is the number of pieces produced by the cutter per grind. This practice has been found to be sound.

Table III gives highly conservative cutting speeds for milling with sintered carbide-tipped cutters, based on the hardness of the material cut.

Feeds and Depths of Cut. The feed rate is the product of the chip load, the number of teeth in the cutter, and the number of revolutions per minute the cutter makes. This was expressed in Chapter XIII, by the formula, $F = fTN$. The formula for the chip load was given as $f = \frac{F}{TN}$. Modern milling machines have a dial on which the feed rates are indicated. The feed rate is selected on the basis of job requirements and the machine then set accordingly.

The chip load can be computed if the r.p.m. of the cutter and the number of teeth are known. For example, assume it is desired to know the feed rate when a 6" milling cutter with 8 carbide-tipped teeth is machining various widths of cut at depths of 1/16", 1/8", and 1/4". It is operating at a speed of 500 f.p.m. and is powered by a motor of

TABLE III. CONSERVATIVE SPEEDS FOR MILLING WITH SINTERED CARBIDE CUTTERS UNDER AVERAGE SHOP CONDITIONS

Material to Be Milled	Brinell Hardness	Cutting Speed in Feet per Minute	
		Roughing 1/8" to 3/16" Depth of Cut	Finishing 1/16" or Less Depth of Cut
Cast iron			
Soft	120-200	275-325	325-375
Medium	200-280	225-275	275-300
Hard	280-400	175-225	225-250
Chilled	400-600	50-100	75-150
Malleable cast iron			
Soft (40,000 to 50,000 tensile strength)	100-140	300-375	375-450
Medium (50,000 to 70,000 tensile strength)	100-140	250-300	300-375
Hard (70,000 to 100,000 tensile strength)	100-140	200-250	250-300
Cast steel			
Soft	110-160	150-250	250-325
Medium	161-250	125-200	200-275
Steel			
Soft	up to 200	175-250	250-350
Medium soft	201-275	150-225	225-300
Medium	276-350	100-150	150-200
Medium hard	351-425	70-120	100-150
Hard	426 and up	30-50	70-100
Brass	140-180	250-350	350-500
Bronze	140-200	250-350	350-500
Aluminum	21-118	1,000-1,500	1,500-2,000
Aluminum alloys	45-150	700-1,000	1,000-1,500

7 1/2 hp. The factor K may be considered at 1. A series of computations can now be made by substituting the known and assumed values in the formula

$$f = \frac{hp}{KdNTw}$$

which can be modified to give the feed rate directly, becoming

$$fNT = \frac{hp}{Kdw}$$

TABLE IV. FEED RATES FOR A 6" CUTTER WITH 8 TEETH, CUTTING SOFT STEEL IN A 7.5 HP MACHINE AT A CUTTING SPEED OF 500 F.P.M.

Width of Cut in Inches	Feed Rates and Chip Loads					
	Feed Rate 1/16" Depth	Chip Load	Feed Rate 1/8" Depth	Chip Load	Feed Rate 1/4" Depth	Chip Load
1	120	.047	60	.0236	30	.0118
1 1/2.....	80	.0314	40	.0157	20	.0078
2	60	.0236	30	.0118	15	.0059
2 1/2.....	48	.0188	24	.0094	12	.0047
3	40	.0157	20	.0078	10	.0039
3 1/2.....	34.3	.0135	17.15	.0067	8.57	.0036
4	30	.0119	15	.0059	7.5	.0039
4 1/2.....	26.7	.0105	13.35	.0052	6.67	.0026
5	24	.0094	12	.0047	6	.0024

Since fNT = F, then

$$F = \frac{hp}{Kdw}$$

The application of this formula can be checked by using it to solve the machining problem described in the last paragraph. The known values are d = 1/16" or .0625, D = 6, hp = 7.5, T = 8, K = 1, w = 4 (assumed), and s = 500. N is found by substitution in the formula

$$N = \frac{S \times 12}{3.14 \times D}$$

$$N = \frac{500 \times 12}{3.14 \times 6} = 318 \text{ r.p.m.}$$

The values are now substituted for the unknowns in $F = \frac{hp}{Kdw}$, resulting in

$$F = \frac{7.5}{1 \times .0625 \times 4} = 30'' \text{ per minute}$$

From this, the chip load can be computed by substitution as follows:

$$f = \frac{F}{NT} \text{ or } f = \frac{30}{318 \times 8} = .0118''$$

The values can now be computed for all widths of cut and arranged

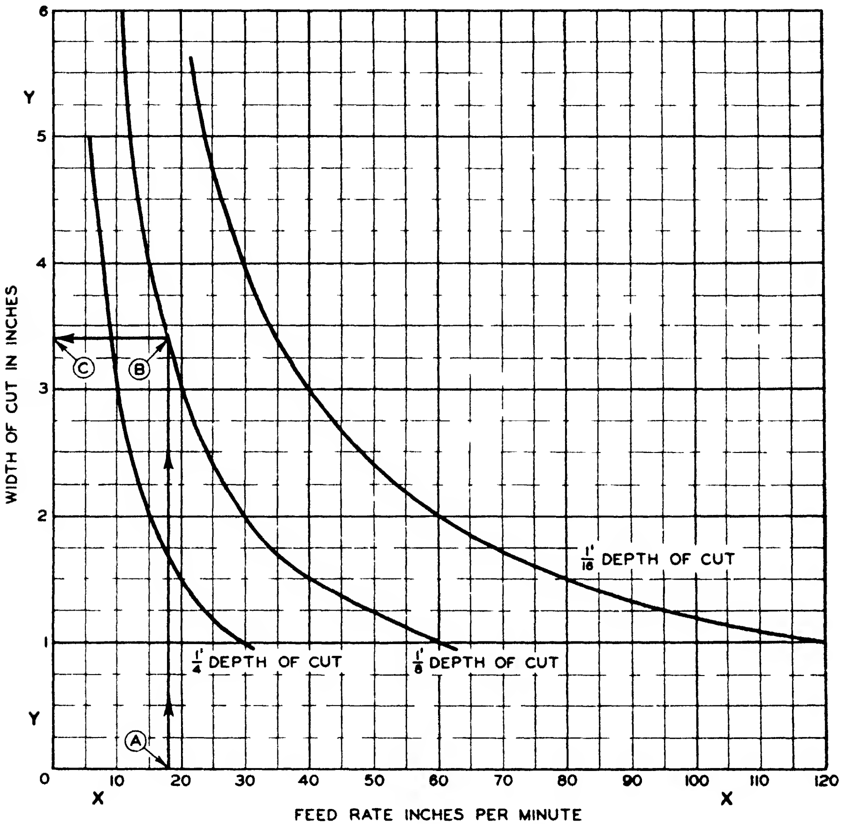


Fig 1. Depth of Cut Plotted against Feed Rates and Widths of Cut as Given in Table IV

in tabular form as illustrated in Table IV. They may then be filed away for use as needed.

The values of feed rate and the width of cut from Table IV can be plotted on co-ordinate paper as shown in Fig. 1, where the feed rate is on the X axis, and the width of cut that can be taken with a given feed, together with the depth of cut for a given speed, is plotted on the Y axis. Three curves are shown, but others could be computed and drawn on the same sheet. The advantage of having the data plotted as shown is that feed rates different from those computed can be taken from the curve values of the widths. For example, for the 17.5" feed rate and the 1/8" depth of cut, the width of cut can be 3.4". The reading is obtained by locating the feed rate specified at the bottom of the graph. A line is then extended vertically (in this case, from A) until it intersects the curved line indicating 1/8" depth of cut. At the point of intersection the

line is carried horizontally to the Y axis where the width of cut can be read. The use of graphs in this respect is particularly desirable since the feed rate of any machine is limited to certain values. For example, a Milwaukee milling machine made by Kearney and Trecker has feed rates available of 1/4, 5/16, 3/8, 7/16, 1/2, 5/8, 3/4, 7/8, 1, 1 1/4, 1 1/2, 1 3/4, 2, 2 1/2, 3, 3 1/2, 4 1/4, 5 1/8, 6 1/8, 7 1/4, 8 3/4, 10 1/2, 12 1/2, 15, 17 1/2, 21, 25, 30, 35, 42, 50, and 60'' per minute.

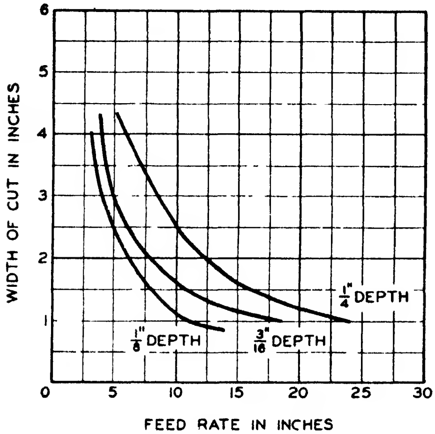


Fig. 2. Depth of Cut Plotted against Feed Rates and Widths of Cut, as Given in Table V, for a Machine Having Less Horsepower Than That in Fig. 1

For machines powered with smaller motors, the curves shown in Fig. 1 will come closer to the Y axis when the same depth of cut is used for computation. This is shown in Fig. 2, which is plotted from data in Table V for a 4'' cutter having 6 teeth, cutting steel of 250 Brinell hardness at 300 f.p.m., constant K being equal to 1.65.

Construction of Cutters. A cutter which is widely used for slot milling, for finishing sides, and for surfacing narrow work is shown in Fig. 3. The cutter consists of a body which is either forged or cast, having teeth tipped with carbide blades. These tips are brazed to the faces of the teeth. Such cutters are made in sizes from 3'' to 8'' in diameter with 6 to 12 teeth, and in widths as listed in Table VI.

In Fig. 4 is shown the typical construction of the teeth found on the carbide milling cutter. It should be noted that the carbide tips project or overlap the body of the tool by 1/32''. This provides free clearance

TABLE V. FEED RATES AND WIDTH OF CUT FOR A 4'', 6 TOOTH CUTTER IN A 5 HP MACHINE

Width of Cut in Inches	Feed Rates for Depth of Cut		
	1/8	3/16	1/4
1	24.25	16.2	12.1
1 1/2	16.15	10.75	8.05
2	12.12	8.1	6.05
2 1/2	9.7	6.46	4.84
3	8.1	5.4	4.03
3 1/2	6.9	4.62	3.45
4	6.05	4.05	3.03

behind the teeth and permits resharpening of the cutter without grinding the body unless, of course, the teeth have been worn down below the overlap. The clearances and relief angles are indicated in Fig. 4.

Carbide-tipped, side milling cutters of the type shown in Fig. 4 are used for surfacing work, for grooving, and for side milling. The teeth

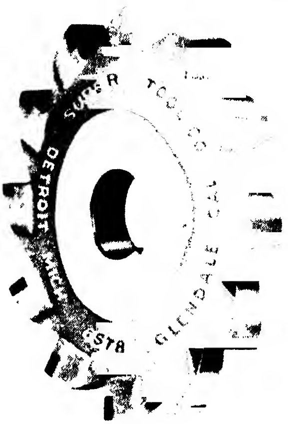


Fig. 3. A Plain, Carbide-Tipped Milling Cutter Widely Used for Slot Milling
Courtesy of the Super Tool Co.

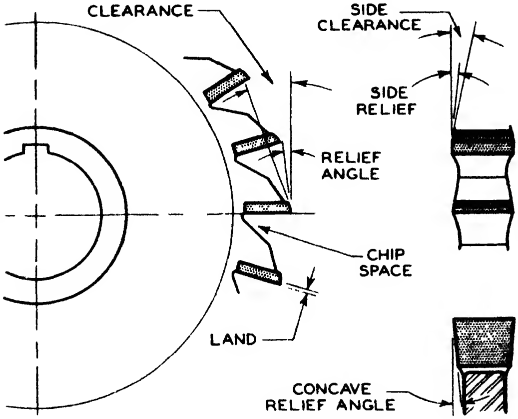


Fig. 4. Tooth Construction and Nomenclature of the Side or Slot Milling Cutter

of the cutter have a relief angle on the periphery so as to obviate any rubbing action of the surface of the tooth against the work. This relief angle is from 3° to 5°, sometimes more, depending on the material being machined, and extends from the cutting edge for the width of the land. Behind the land there is a clearance of about 10°. The purpose of the clearance is to prevent the heel of the tooth from rubbing on the work, and to reduce the amount of material necessary for removal when resharpening the cutter.

TABLE VI. SPECIFICATIONS FOR CARBIDE-TIPPED PLAIN MILLING CUTTERS

Cutter Diameter	Widths in Inches	Arbor Hole in Inches	No. of Teeth
3	1/4, 5/16, 3/8, 7/16, 1/2	1	6
4	1/4, 5/16, 3/8, 7/16, 1/2, 9/16, 5/8, 3/4, and 7/8	1 to 1 1/4	8
5	7/16, 1/2, 9/16, 5/8, 3/4, 1	1 to 1 1/4	10
6	1/2, 5/8, 3/4, 1	1 to 1 1/4	12
7	3/4, 1	1 to 1 1/4	12
8	3/4, 1	1 1/4 to 1 1/2	12

Cutter teeth, when used for side milling, have a side relief of from 2° to 3° and a secondary clearance of 10° to 12°. The relief is on the land part of the side of the tooth. Its function is to prevent the tooth

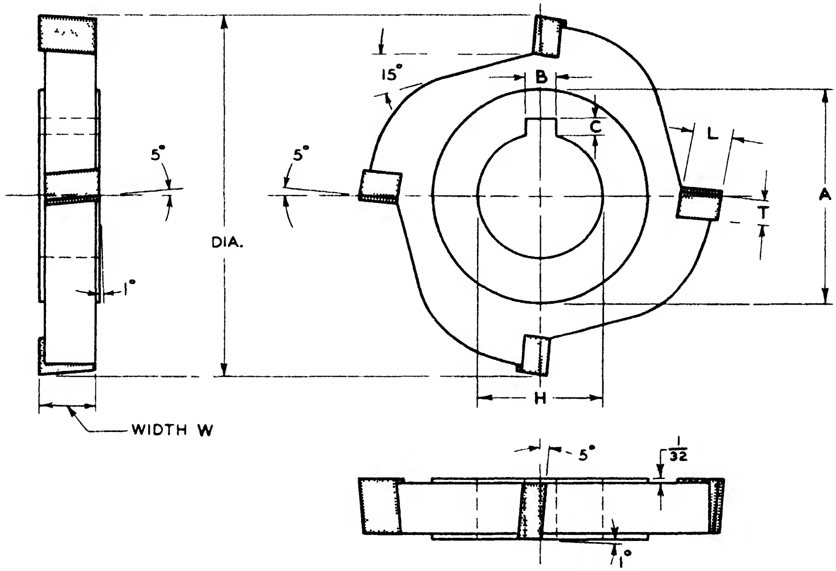


Fig. 5. Details of the Staggered-Tooth, Slot Milling Cutter, Specifications of Which Are Given in Table VII

from dragging on the work when cutting on the side or in a slot, and to present a keen cutting edge to the work. The sides of the teeth are concave toward the center by 1/2° to 1°. This concavity prevents the sides of the teeth from dragging on the sides of the cut, eliminating the frictional losses of power and the heat that is generated when rubbing takes place. When the cutter is reground, it narrows down somewhat, but the amount of narrowing is not great because the angle of concavity is so, small.

The thickness of the carbide tips varies between 3/32'' and 1/4'', depending upon the size of the cutter, the number of teeth, and the severity of the work it is expected to perform. In general, the thickness of the tip is not as important as its attachment. If brazing is used and has been done properly, the thickness of the carbide may be less. The carbide should be more substantial if for poorly brazed teeth, but it need never exceed 1/4'' for any type of cutter.

Cutters of this type may have straight teeth or teeth which are helically inclined to the axis. The teeth can be radial, and can have a positive or a negative radial rake. For machining aluminum and magnesium, the radial rake angle may be positive. That is, it will have a hook. For cast iron, the angle is zero, or it is said that the teeth are radial.

TABLE VII. SPECIFICATIONS FOR CARBIDE-TIPPED SLOTTING MILLS

Cutter Diameter	Number of Teeth	Hole Diameter	Hub Diameter	Widths of Cutters	Blanks	
					Thickness	Length
3	4	1	1 1/4	1/4, 5/16, 3/8, 7/16, 1/2	3/16	5/16
4	4	1 1/4	1 1/4	1/4, 5/16, 3/8, 7/16, 1/2	3/16	5/16
5	6	1 1/4	1 3/4	7/16, 1/2, 9/16, 5/8, 3/4	3/16	3/8
6	6	1 1/4	2	1/2, 5/8, 3/4 7/8, 1	1/4	7/16
7	8	1 1/2	2	3/4, 7/8, 1	1/4	1/2
8	8	1 1/2	2	3/4, 7/8, 1, 1 1/4	1/4	1/2

For cutting steel, the radial angle is made negative from 5° to 10° and the axial angle of the tooth is also from 5° to 10° negative.

Such cutters may have teeth running alternately in opposite directions on the periphery. That is, the axial angle of one tooth may be positive and that of the next, negative. When teeth are so arranged, they are said to be staggered. Many tool engineers prefer tools of this type.

Slotting with Staggered Teeth. In Fig. 5 is shown a staggered-tooth, slot-milling cutter which is tipped with carbide. The radial and the axial angles of this cutter are 5° negative. Table VII gives specifications for such slotting mills.

Slab Mills. Slab mills are milling cutters which take a wide cut. Such a cutter is shown in the setup pictured in Fig. 6. This tool is of high-speed steel. So far, these cutters have not been successful performers when tipped with sintered carbides. This is not because carbide-tipped slab mills are not suitable for machining operations, but because there are other limiting factors. These are:

1. The difficulty of making the helical blades and attaching them by brazing to the steel or cast-iron body.

2. The nonrigidity of the machine arbor for this type of application, which requires the exertion of considerable force in order to take a wide cut.

3. The lack of power at most machines. Slab milling with sintered carbide requires considerable power to take a wide cut at substantially coarse feeds and high speeds.

A single-helix slab mill was illustrated in the last chapter. This mill was used for experimental purposes to determine various factors pertaining to machining with this type of cutter. Previously, it will be remembered, a 4" slab mill tipped with tantung, one of the cast cobalt-

chromium-tungsten system alloys, was shown in Chapter IV. The blades were cast separately in this construction, then brazed to the teeth which had been milled to the correct helix angle. Tests performed in different

Fig 6 Slab Milling Cutter of High-Speed Steel
Courtesy of Brown & Sharpe Mfg Co

plants on production work with this cutter gave excellent results, indicating there is good reason to believe that slab milling with carbide-tipped cutters awaits only the development of better supports for the cutter in operation, more powerful milling machines, and properly designed cutters.

Overlapping Staggered Teeth. Another type of slab milling cutter has teeth staggered so that the surface worked is not engaged at once with the whole width of the cutter tooth, but with slightly more than half of it. The teeth have a slightly greater width than half that of the cutter, being tipped with 3/16" × 3/8" × 7/8" blanks. These cutters are made with 5° negative radial and 10° negative axial angle for steel cutting, and with 7° positive rake and 15° positive axial rake angle for

milling cast iron and nonferrous metals. The general design of the tool is shown in Fig. 7.

Inserted Blade Plain Mills. Plain and side mills are generally made with solid teeth up to 8'' in outside diameter. Mills having a greater diameter usually are made with inserted blades, although smaller diameter cutters also may have this type of construction. In Fig. 8 is shown a 3 1/4'' diameter milling cutter with 8 inserted blades. It is designed for light surfacing of work. The cutter consists of a steel body, A, having milled slots, B, to receive sintered carbide or cast alloy blades, C. The blades are then locked in position by pressure exerted by the taper pins, D, which tend to spread the body of the cutter from the split at E. The cutter, having but eight teeth, needs only four taper pins to lock the blades in position, pressure being exerted by each pin against both adjacent teeth.

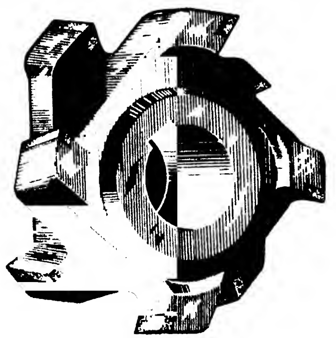


Fig. 7. A Carbide-Tipped, Overlapping, Staggered-Tooth Cutter Used for Slab Milling

The carbide-tipped cutter blades, shown in detail at the bottom of Fig. 8, are tipped with sintered carbide blanks. The blades are accurately made and should have a free, sliding fit in the slots in order that they may be located and locked into place without difficulty. Solid carbide blades 5/16'' x 11/16'' x 1'' could also be used in this design. However, the cutter body would then have to be wider to provide better backing for the carbide. The overlap of the blades would have to be reduced from 1/32'' to 1/16'' on each side.

The cutter blades are set to provide a 5° radial rake angle to the teeth. The axial rake is zero. The teeth, after being locked in place, are ground on the periphery to 10° relief behind the front cutting edges and with a secondary clearance of 20°. Such angles are frequently used when milling aluminum or magnesium, although a bigger front rake angle would be more desirable and would produce a better finish on the work.

Face Milling. The greatest development in milling with the carbides has taken place in face milling. This process entails the removal of material by a rotary cutter mounted on the milling machine spindle so that the cutter revolves with the spindle of the machine either in a vertical or a horizontal plane. In some applications, it revolves in an inclined plane, with the axis of rotation always perpendicular to the surface being machined. In this operation, the tooth of the cutter engages the work with less contact than is possible in plain or slab milling, thus requiring less force to do the work.

Two typical face-milling operations are shown in (A) and (B) of Fig. 9, and additional are to be found in various other sections of this book. Face mills may be divided into several classes according to their design.

For the purposes of this discussion, these classes may be listed as follows:

1. Face mills with carbide blanks brazed to the cutter body.
2. Face mills with inserted steel blades tipped with carbide blanks.

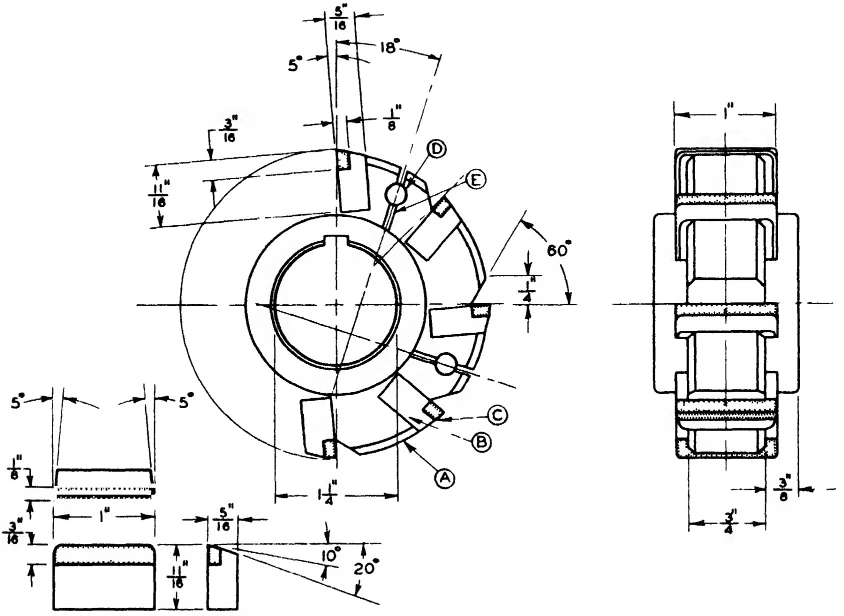


Fig. 8. Details of a Milling Cutter Having Inserted Teeth Tipped with Carbide

3. Face mills with solid, sintered carbide cutting blades.
4. Solid-type end mills.
5. Shell-type end mills.

Face mills having carbide tips brazed directly to the body also have been illustrated in several places in this book. As seen in Fig. 10, the face mill consists of a body made of steel which is usually forged or cast to shape, and carbide tips. The cutter body fits the #50 standard spindle nose and is fastened to the spindle of the machine with 4 fillister head screws.

In general, there are three types of such cutters available, differentiation being based on the radial and axial angles. The angles vary with the material to be cut. For cutting steel, the radial and axial angles are from 5° to 10° negative. For cast iron, they are either zero or 5° positive. For cutting aluminum and magnesium, the angles vary between 10° and 20° positive.

It should be noted that the cutter is provided with ample chip clear-

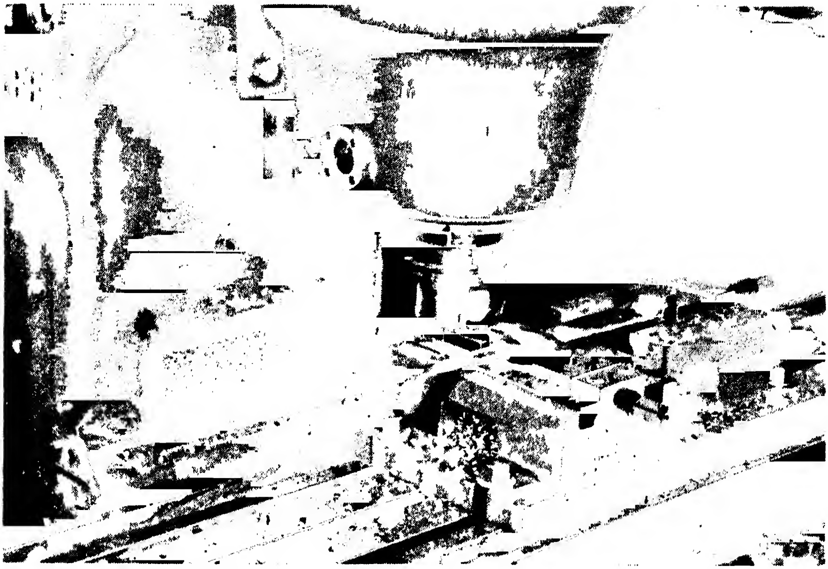


Fig 9(A) Face Milling a Machine Casting with a 4" Cutter Having Negative Rake Teeth
Courtesy of Kearney & Trecker Corp

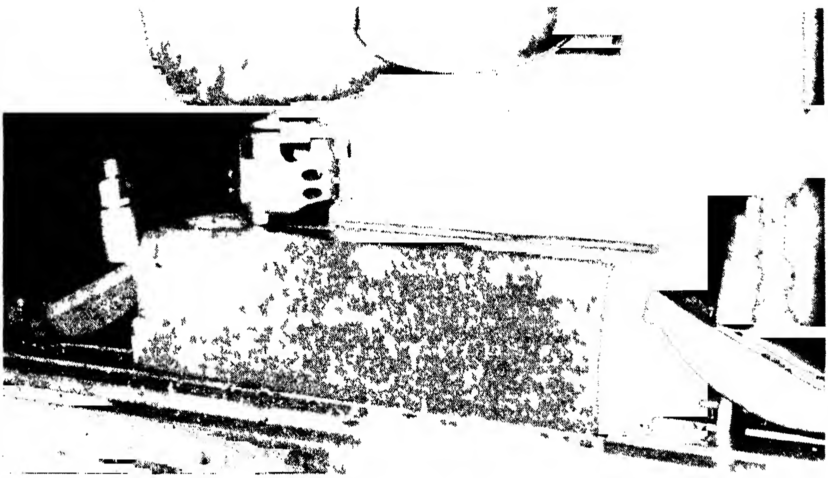


Fig 9(B) A "Step Mill," Used Here as a Face Mill to Cut Stainless Steel at 860 f p m with a Chip Load of 010"

ance, a point which is of paramount importance to its proper functioning, and one which is frequently overlooked by designers and tool makers. Face mills of this type are of rugged construction and are often made heavy, as in Fig. 10, to give a flywheel effect. The number of teeth is usually equal to the diameter of the cutter in inches, or to the diameter plus two.

Shape of Cutter Teeth. Fig. 11 illustrates the general shape to which the cutter teeth are finished. At (A) is shown the tip which is ground to a square corner when the cut is against a square corner. This type of cutter corresponds to the single-point tool without the side cutting-edge angle, required for cutting against square corners, which was described in earlier chapters. At (B) is shown the tip of a cutter which has a bevel, or an entrance angle other than zero, corresponding to the side cutting-edge angle in the single-point tool. This angle is usually about 35°. A cutter of this type is used for roughing out work and usually assures long tool life. At (C) is shown the shape commonly used when commercially good finish is required. It has a land on the face equal to 1/16" to 1/8", on which the concavity may be from zero to 3/4 of one degree. At (D) is shown the type of cutter tooth which has a bevel angle the same as the tips shown in (B) and (C). In addition to this, it has a small lead angle of 3/4° to 1° extending about 1/16" in front of the land. This tooth shape is used where there is a tendency for chattering.

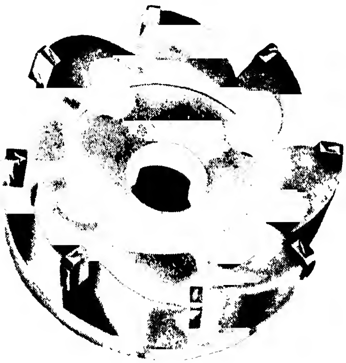


Fig. 10. A Typical Face Mill with Brazed-in Carbide Tips
Courtesy of the Super Tool Company

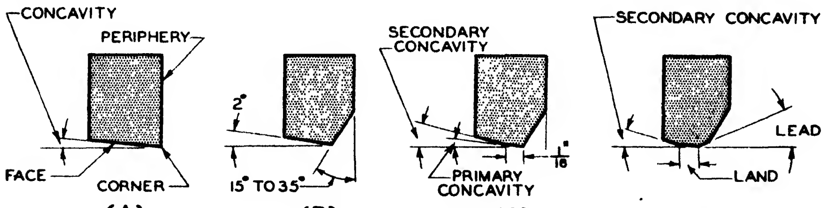


Fig. 11. Details of Four Shapes of Milling Cutter Teeth

Primary Relief Angles. For best performances, the primary relief angle behind the cutting edges of carbide tipped cutters is not the same for all materials cut. Table VIII gives the values of primary relief angles on the periphery of the cutter, the corner, and the face, for face milling or slotting aluminum, magnesium, cast iron, and steel.

Inserted Blade Face Mills. Perhaps the most commonly used face mills are of the inserted steel blade type. The blade inserts are tipped with the grade of carbide most suitable for the job at hand. This construction is used for reasons of economy, and because it simplifies the manufacture of the cutters. The steel blades are much easier to fabricate to size and shape than the blades of solid carbide, although the latter are gaining in favor among many engineers. On the other hand, brazing of the carbide blanks to the steel blades, followed by grinding them to the required dimensions, is a source of some difficulty. However, the ease with which these blades can be readjusted for grinding and the facility with which they may be replaced when necessary, causes many tool engineers to design or select cutters of this type to solve their milling problems.

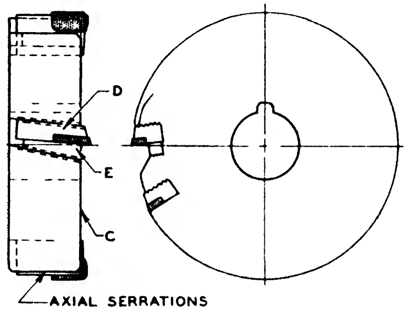


Fig. 12. The Goddard & Goddard Type of Face Mill with Inserted Carbide Tipped Blades

Fig. 12 shows a Goddard and Goddard shell type face mill with inserted steel blades tipped with carbide. This cutter consists of a body, C, a serrated steel blade, D, and a serrated wedge, E. The body of the cutter has slots in the periphery, each with its side at the proper rake and axial angle. The slot is narrower toward the rear of the cutter to effect a locking action of the wedge on the blade. The sides of the groove are serrated. The wall of the groove on the blade side has serrations either straight or inclined as shown in Fig. 13. The serrations on the wedge side are parallel to the bottom of the groove. Serrations on the blade and in the slot prevent the blade from shifting under the impact of the cutting forces, while the serrations on the side prevent it from climbing.

TABLE VIII. PRIMARY RELIEF ANGLES FOR FACE-, SIDE-, AND END-MILL CUTTERS

Material Cut	Periphery Relief in Degrees	Corner Relief in Degrees	Face Relief in Degrees
Aluminum........	10	10	10
Cast iron.........	7	7	5
Steel...............	4-6	4-6	3-4

To adjust the blades for grinding, it is necessary to drive out the wedges, then advance each blade one or more notches or serrations. The advancing is done in the axial direction. Following this, the blades

are locked in place by driving in the wedges again. The serrations are spaced 1/16'' apart. The blades are standardized and are built with sufficient accuracy so that replacement can easily be made when either

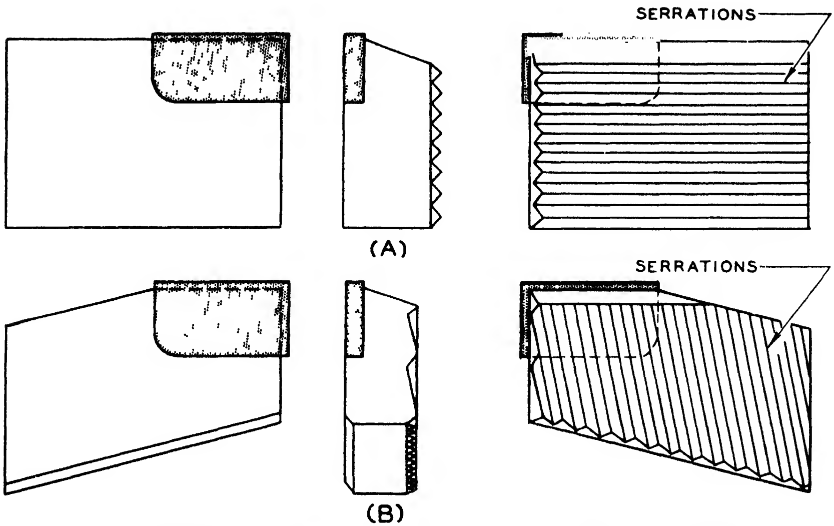


Fig. 13. Straight-Type Blades for the Goddard Milling Cutter (A) and Blades for the Cone-Type Goddard Milling Cutter (B)

the carbide has been used up or when a blade has been accidentally damaged. Such cutters are made either for right- or left-hand operation with radial and axial angles suitable for cutting steel, cast iron, or the nonferrous metals.

Another type of face mill, using the same type of blade as shown in Fig. 13, is illustrated in Fig. 14 at (A). This cutter is known as the cone-type, carbide-tipped face mill and is similar to the cutter in Fig. 12. The body of the cone-type cutter is conical, as its name suggests, and the blades are placed on it at an angle.

Still another type of inserted-tooth face mill tipped with carbide is shown in Fig. 14 at (B). This cutter consists of a body with triangularly shaped bits tipped with carbide which are inserted in triangular holes broached in the body of the cutter. The bits are locked in place with a simple lock screw, while another screw serves as backing for the cutter and also to adjust the bit. The bits are either 5/8'' or 7/8'' triangles. The smaller size bit is used for cutters which are intended for cuts up to 3/16'' in depth, while the larger size is used when the cut taken is greater than 3/16'' in depth. As is seen in the illustration, the cutter has ample chip clearance, which is advantageous when heavy cuts are taken. The cutter has fewer teeth for heavy-duty work than for lighter work.

Adjustment of the blade in this type of cutter consists of loosening

the clamping screw and turning the adjusting screw so as to move the bit either forward or backward. This is shown clearly in (B) of Fig. 14. Removal of the bits is unnecessary except when replacement is neces-

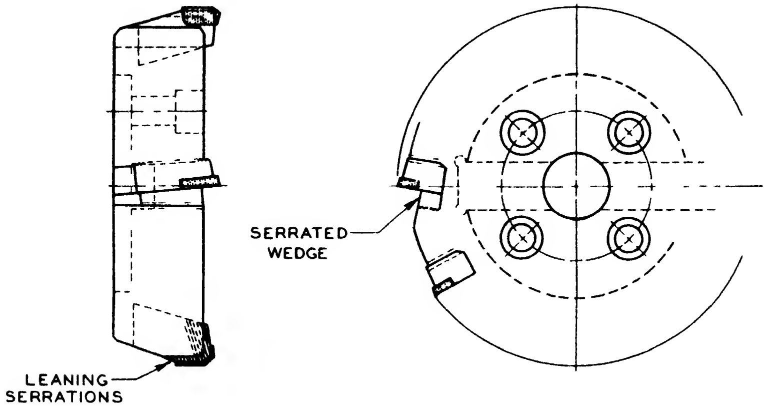


Fig. 14.(A) The Goddard, Cone-Type, Carbide-Tipped Face Mill Which Uses the Blade Shown in (B) of Fig. 13

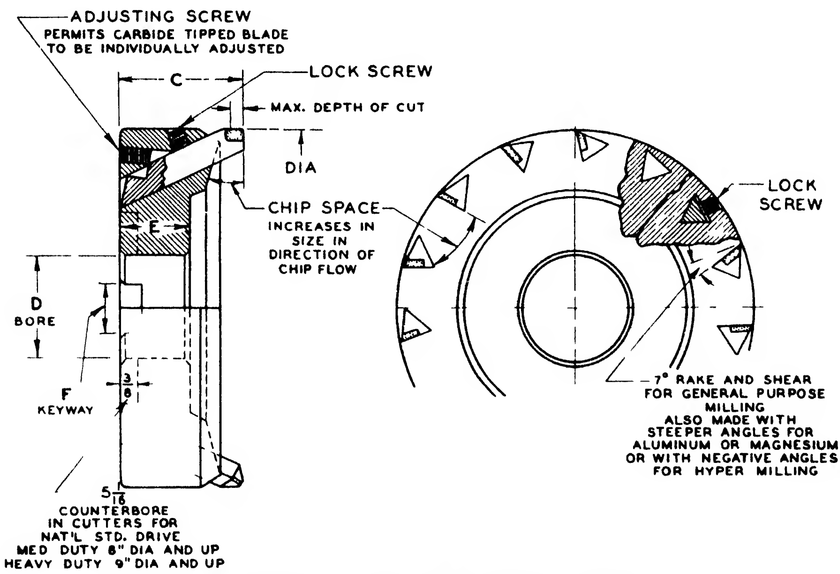


Fig. 14.(B) A Cone-Type, "Tri-Bit" Face Mill Made by Weddell Tools, Inc.

sary. The tools are made to cut steel, cast iron, or nonferrous metals. They are furnished with varying radial and axial angles for use on various materials.

Another design of carbide-tipped, inserted-tooth face mill, a drawing of which is shown in Fig. 15, has a steel, forged body into which are inserted blades made of round steel, machined flat on one side and serrated as shown in A, Fig. 16. There is a semicylindrical shoe B, which completes the cylinder of the blade form. The shoe has serrations to fit against the blade and is locked in position by a wedge made of a pin, D, which is machined flat on one side so as to fit well against the flat surface, C, in the shoe.

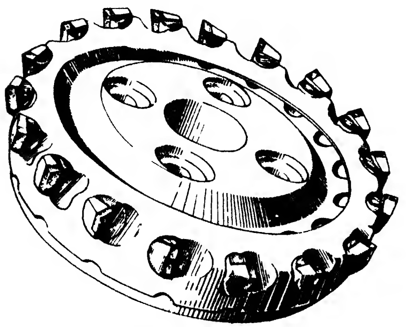


Fig. 15. A Heavy-Duty, Inserted-Blade Face Mill
Courtesy of the Lovejoy Tool Company

These heavy-duty, Lovejoy cutters are designed to take a cut 3/4'' deep. They are made for cutting cast iron, steel, and nonferrous metals simply by varying the radial and axial angles. They are provided with liberal chip clearance, and the teeth project far enough from the body so that several resharpenings are possible before any adjustment of the blades is necessary. Special instructions for blade adjustment are furnished by the manufacturer for each user.

A design of this kind permits the removal of every other tooth from the cutter for use on machines which do not have enough power for speeds and feeds at which carbide-tipped cutters are most economical. How this works in actual practice can be illustrated by an example:

Assume a 10'' heavy-duty milling cutter having 16 teeth is available for milling 6'' steel forgings of 200 Brinell hardness at a cut depth of .375'', chip load of .008'' per tooth, and cutting speed of 300 f.p.m. It is desired to know how many teeth would have to be removed in order to use the cutter in a machine powered with one 25 hp motor. The known values are d = .375, f = .008, w = 6. The unknown values are K, which can be taken from Table IV in Chapter XIII and is found to be 1.55, and N, the revolutions per minute. This can be found by substituting the known values in the formula

$$N = \frac{S \times 12}{3.14 \times D}$$

giving

$$N = \frac{300 \times 12}{3.14 \times 10} = 115 \text{ r.p.m.}$$

The known values are now substituted in the formula given earlier for determining teeth number, resulting in

$$T = \frac{25}{1.55 \times .037 \times .008 \times 115 \times 6} = 7.8 \text{ or approximately } 8.$$

When teeth are removed from a cutter, there should be equal spaces left between them, or the operation will become "jumpy." Most of the cutters described can be used with some of the blades removed when

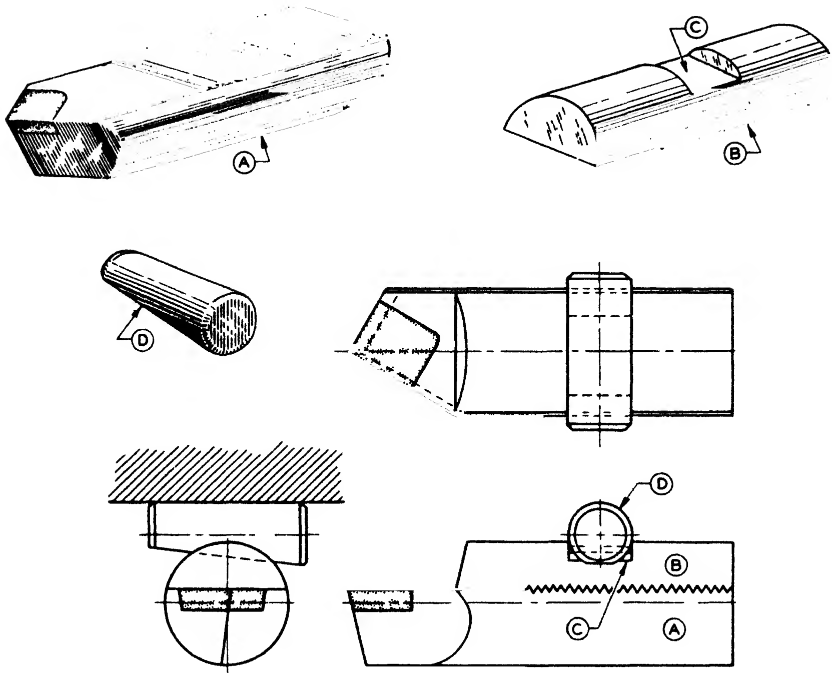


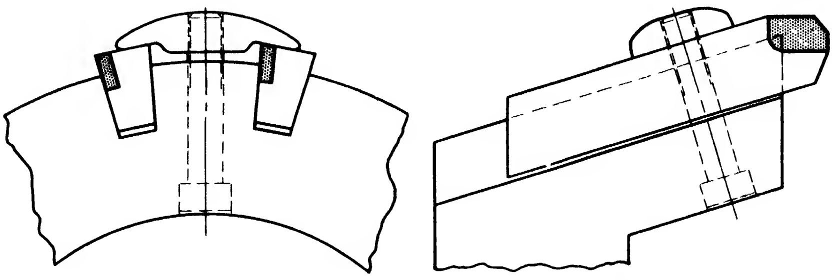
Fig. 16. Blade Details for the Lovejoy, Heavy-Duty, Inserted-Blade Face Mill
Courtesy of the Lovejoy Tool Company

conditions call for cutting with fewer teeth. It will be obvious, of course, that those with blades wedged or locked in pairs may not always be reducible to fit the exact number of teeth called for under the conditions, but in most cases, the number can be reduced somewhere close to the optimum.

Another type of face-milling cutter using radially tapered blades tipped with carbide is the Continental, shown in a number of places in this book. The blades of this cutter are made accurately to fit into slots which have been milled in the periphery of the cutter body. The body itself is hardened and the slots are ground after hardening to remove or correct any distortion that might have occurred during the operation. The blades are then set in the slots, protruding the required distance from the body, and are fastened securely with clamps actuated by Allen-type cap screws.

The details of the construction of this cutter are shown at (A) in Fig. 17, where the blades are in position with the Allen screw tightening

the clamps which, in turn, force the blades into the angular slots. In this construction there must be ample clearance at the bottom of the blade to assure a good seat throughout the life of the tool. Obviously,



(A) (B)

Fig. 17. Details of the Continental, Inserted-Blade, Face Milling Cutter

the angular slots in the body should be made with sufficient accuracy to assure a good, solid seating for the blades. The general shape of the carbide-tipped blade is shown in (B) of Fig. 17.

When it becomes necessary to move the blades forward for resharpening, the clamps are loosened and the blades freed by tapping with some soft material. When the blades have been moved the required amount, they are clamped firmly in place. At present, there are three types of cutters of this kind. Their essential difference is in the mounting, a matter that will be discussed in detail later in this chapter.

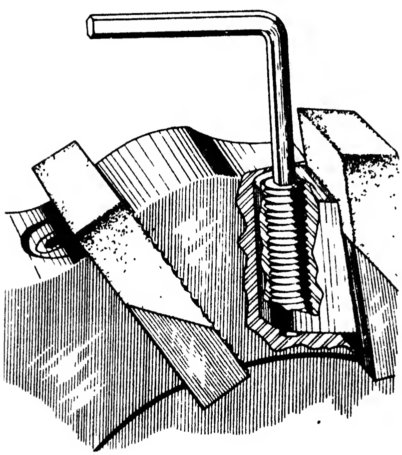


Fig. 18. The "Jack-Lock" Wedge Method of Retaining Carbide-Tipped Milling Teeth
Courtesy of the McCrosky Tool Corp.

A heavy-duty, face-milling cutter with serrated steel blades tipped with sintered carbide and locked in place by means of a special device, is the McCrosky "Jack Lock," a cutaway drawing of which is presented in Fig. 18. This cutter consists of a conical body, carbide-tipped, serrated blades, wedges for locking the blades in place, and fine pitch screws which operate the wedges for locking and unlocking. The cutters are designed to fit directly on the nose of the national standardized spindles. They have a 4" bolt circle with 1 3/4" diameter holes. Cutters to fit other spindles than the standard are made special.

The cutters are designed to take heavy cut in relatively rugged service.

As can be seen from the illustration, the wedge of the McCrosky "Jack Lock" mechanism is semicylindrical in cross section and fits into a similarly shaped recess in the cutter body. Together with the fine pitch, socket-type set screw, it forms an independent unit. In setting the blade or clamping it in place, the set screw is tightened. This action lifts the wedge, forcing it against the side of the recess and the blade. The wedging action forces the serrations on the back of the blade into rigid engagement with the serrations on the body of the cutter, locking the blades securely so they cannot shift under the impact of cutting forces or become loose as a result of vibration.

Solid Carbide Cutting Blades. Some designers prefer milling cutters with solid sintered carbide cutting blades. Their preference is based on the assumption that solid carbide blades, when wedged into the body of the milling cutter, are less likely to crack in operation than are tips which have been brazed onto steel blades. Such cracking is blamed by these adherents on so-called thermal stresses. Whether this is actually so still remains a controversial point, but the assumption that the brazed tips are likely to crack in operation has some justification because the coefficient of expansion of carbide and the steel shank is not the same, that of the steel being approximately twice that of the carbide. Therefore, when cutting is done and, as a result of it, heat is generated, the expansion of the carbide tip is less than that of the steel body to which the carbide is brazed. As a result, a stress is set up in the carbide and in the steel shank. The greater expansion of the steel causes tensile stress (pulling stress) in the carbide. Since carbide is low in tensile stress, it may crack when heat is developed unless, of course, the brazed joint is able to absorb such stresses. In spite of all arguments, properly designed and constructed cutters with tips brazed directly to the body or to the steel blades have proved to be highly satisfactory cutting tools. However, in the case of the solid carbide blades, there can be no expansion stress except those due to uneven expansion of carbide itself.

In Fig. 19 is shown one of the basic wedging principles used in the construction of face mills equipped with solid carbide wedged-in blades. The cutter consists of steel or a special cast-iron body, A, the wedges, B, the carbide blades, C, and the wedge clamping screw, D. The slot in the cutter body may be milled at an angle for the carbide blade, C, and at an angle for the wedge, B.

In setting the blades, the wedge is drawn down by Allen type set screw D, against the carbide and against the side of the slot in the body, causing the blade to be locked securely in position.

The teeth of the wedged-in, solid carbide cutters may have any desired combination of radial and axial rake angles. The cutter illustrated in Fig. 19 has a zero radial and zero axial rake angle, a combination suitable for cutting cast iron. For machining aluminum, both angles may be positive. For machining steel, the angles may be negative.

It should be noted that the cutter has liberal chip space ahead of the cutting edge and a contour which permits the chip to slide out from the face of the tooth. Should the chips wedge into the chip space, trouble is encountered. The machine will either stall or broken blades will result. The blades, to be effective, should be either 1/4'' x 1 5/8'' x 3/4'', 5/16'' x 1 7/8'' x 1'', or 3/8'' x 5/8'' x 1 7/8''.

The Kearney & Trecker Corporation makes a 10'' face milling cutter with 12 solid cemented carbide cutting blades having 7° negative radial and axial angles. The blades are held against the side and the bottom of

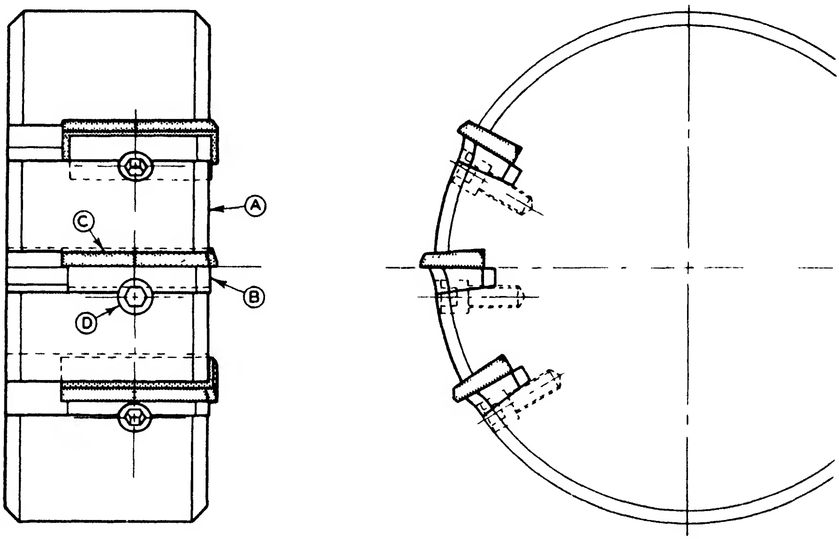


Fig. 19. A Typical Milling Cutter Illustrating a Basic Method of Wedging Solid Carbide Tips in Place

the slot by wedges which are secured by screws. The carbide blanks also are of wedge form. Another Kearney & Trecker solid carbide 10'' face milling cutter with 12 cutting blades has the blades set radially at 15° positive while the axial angle is negative. The faces of the blades are ground 10° negative for a width of .025'' from the cutting edge. The wedges are back of the cutting blades, thus giving a construction in which the blades do not interfere with the chip space and the free flow of chips.

This combination of positive radial rake angle with a negative ground face of .025'' gives very good performance with less expenditure of power than would be possible with a negative radial-rake angle cutter. As will be recalled, the question of power required for a positive rake angle as against a negative rake tool was discussed in the preceding chapter. The Kearney & Trecker cutter provides the advantage of less

power consumption because of the positive rake angle, the longer tool life, and smoother surface finish due to the .025 negative land on the face of the teeth.

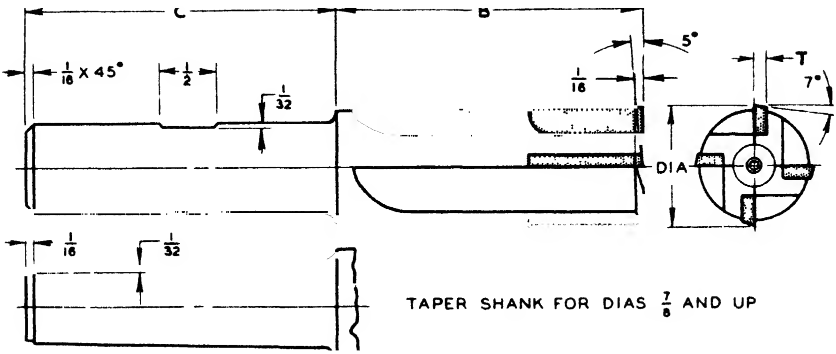


Fig. 20. A Small, Shank-Type, Four-Fluted End Mill with Brazed-on Carbide Tips

The "Jack-Lock" construction previously described also lends itself to the application of solid carbide blades for teeth having either plain or serrated back surfaces. When such cutter blades are adjusted, it is usually necessary to regrind the cutter. Otherwise, there may be high or low blades in the assembly which will bring about unsatisfactory performance.

Shank-Type End Mills. End mills are milling cutters with teeth on the periphery which remove the material while rotating. They are primarily end-cutting tools similar to face mills from which they differ only in certain minor details of design. Fig. 20 shows a small type of end mill which is used where the work is light and the machine is capable of great speed. The tool consists of the fluted body, B, with its tapered shank, C, and carbide tips, T, which are brazed to the recesses in the flutes. The teeth are ground to the same angular relief and clearances as the larger face mills. The teeth are not adjustable except by rebrazing. For this reason, it may become necessary, in regrinding, to grind the steel body.

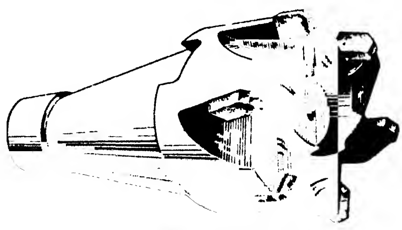


Fig. 21. The Nelco Shank-Type End Mill with Brazed-on Carbide Tips

Another style of end mill of massive, solid design is the Nelco cutter shown in Fig. 21. Again, the carbide tips are brazed directly to the steel body. The mill, having few teeth, has liberal chip space, an item which characterizes good milling cutter design. These cutters are made

with negative radial and axial rakes for milling hard and tough steel on machines where the power is available, with positive radial and negative axial rakes for milling softer steels where power is low, and with posi-

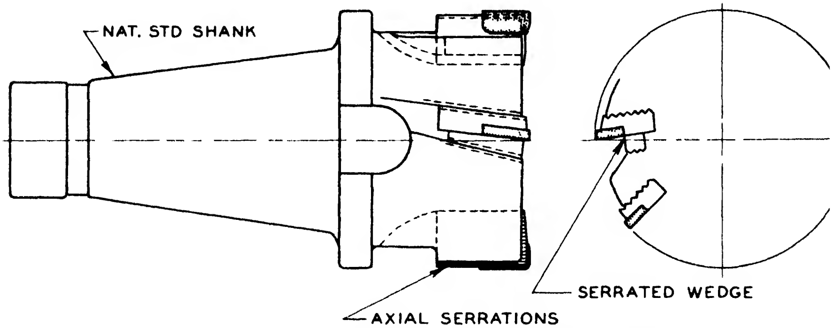


Fig. 22. The Goddard Shank-Type End Mill with Inserted Tipped Blades

tive radial and axial angles for milling cast iron, brass, and bronze. For milling aluminum, duralumin, magnesium, plastics, and zinc base die castings, steep positive radial and axial rake angles are provided.

A straight-shank end mill with inserted carbide-tipped blades is shown in Fig. 22. End mills of this type also may have solid carbide blades wedged in as was the case of the face mills. This end mill,

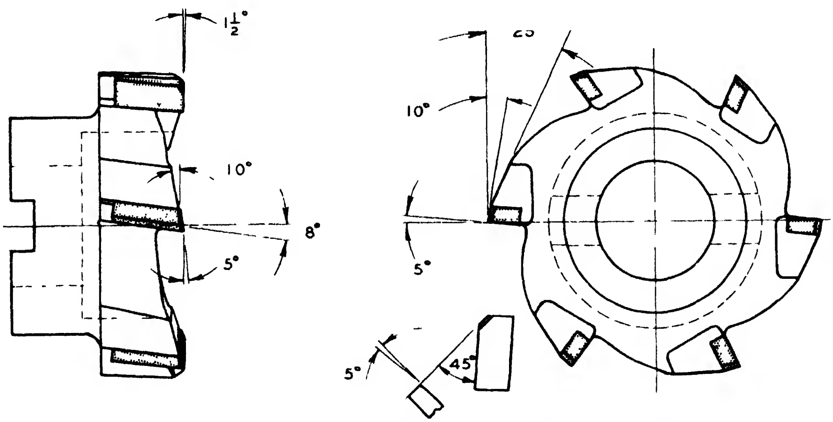


Fig. 23. A Typical Shell-Type End Mill with Brazed-on Carbide Tips

manufactured by Goddard and Goddard, consists of a body having a tapered shank which fits into the spindle of the milling machine. It should be noted that the blade inserts are serrated and that matching serrated wedges are provided for locking the blades in position in the

body of the cutter. The cutter shown in the drawing may have positive radial and axial rake angles suitable for cutting cast iron, brass, bronze, aluminum, and magnesium, or it may have negative angles for cutting steel. The cutters are made either for right- or left-hand operation. For efficiency of chip removal, the clearance should be liberal and of a shape that will not interfere with the free flow of the chips.

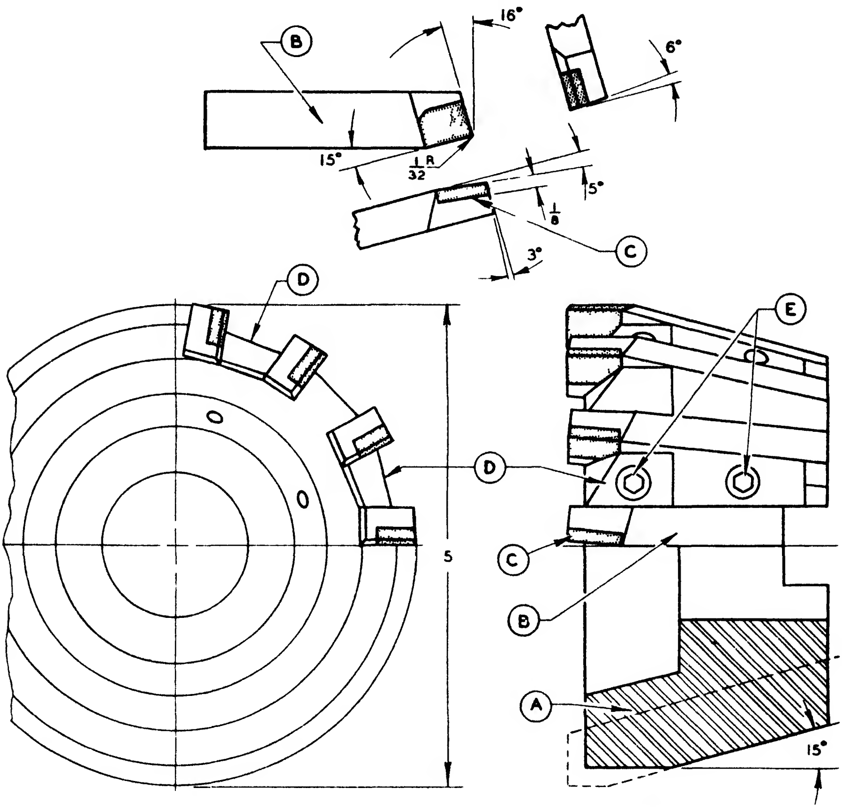


Fig. 24. A Typical Shell-Type End Mill with Inserted Carbide-Tipped Blades

Shell-Type End Mills. More common than the solid-shank end mills are the shell-type end mills which, in reality, are face mills mounted on a special shank which in turn fits into the milling machine spindle. The shell-type face mills may have the carbide tips brazed directly onto the body, they may have the tips brazed to steel blades which in turn are wedged into the body of the mill, or they may have solid carbide blades wedged into the body.

Fig. 23 illustrates a 3 1/2", 6-tooth, shell-type end mill with carbide tips brazed to the body. The tips for this size cutter are 3/16" x 3/8"

x 3/4'' which affords enough material for repeated resharpening. The body is made of hard steel, or of Meehanite cast iron. The cast-iron bodies are preferred because of the great compressive strength they possess, making them capable of resisting the forces imposed upon them by the cutting action. These mills, like the solid-shank types, may have a positive or negative radial and axial rake, depending upon the material that is to be cut.

A typical, inserted blade style of shell end mill is shown in Fig. 24. This cutter consists of a conical body, A, made of heat-treated steel or cast iron; steel blades, B, tipped with carbide blanks, C; the wedges, D; and screws, E. The carbide-tipped blade is shown in detail in the upper section of Fig. 24. The blades are set in slots milled in the body and are fastened by means of wedges. The wedges are activated by the cap screws. The cutter shown in the illustration is made for left-hand operation. With this type of construction, it is possible to provide the cutter with liberal chip clearance. This feature can be seen in bottom view of the cutter at the left of the illustration.

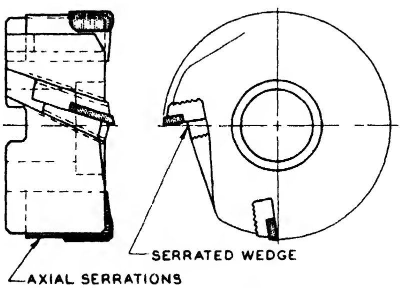


Fig. 25. Shell End Mill Made to Fit on a Tapered Shank

A third type of shell mill is the Goddard and Goddard, shown in Fig. 25. It is similar in most respects to the cutter shown in Fig. 21, but differs in some details of construction, notably that it fits on a tapered shank which is made to fit the spindle. Its chief advantage is a somewhat lower first cost and the fact that as occasion demands, different sizes can be used on the same shank.

Fly Cutters. Milling cutters having usually one or two teeth and mounted or brazed on relatively larger size bodies, are called fly cutters. The cutting action of these tools is the same as that of face or end mills. The chief advantage of the fly cutter is in its low initial cost and a smaller demand for power during its operation. This latter point is true because most of them engage the work only through one part of a revolution. The action of these cutters is intermittent, therefore, but this is offset somewhat by their weight. A disadvantage of the fly cutter is its relative slowness in machining, for, although the cutting speed may be high and the chip load reasonably high, the milling machine table travel is equal to the chip load per tooth. For cutters with two teeth, therefore, the table travel is still only twice the chip load. These cutters are used when the quantity of the work to be done is small, and where machines do not have sufficient power to use cutters with more teeth.

In (A) of Fig. 26 is shown a two-blade fly cutter with a solid shank.

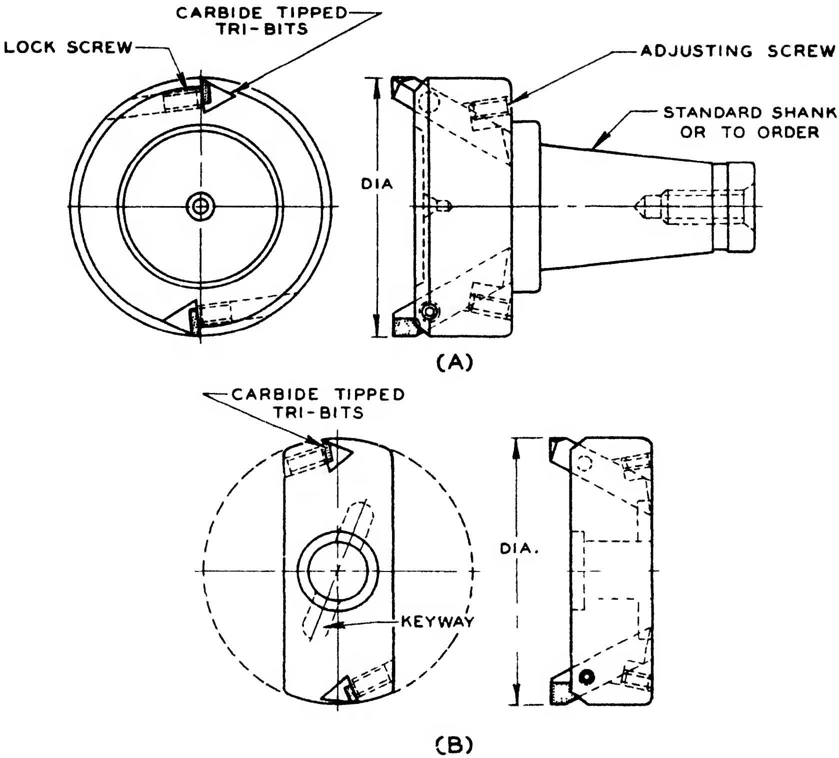


Fig. 26. Two-Blade Weddell Fly Cutter Having a Solid Shank (A) and a Two-Blade Shell-Type Weddell Fly Cutter (B)

It is to be noted that a fly cutter may be of any of the designs discussed in the foregoing paragraphs. Almost all cutter manufacturers supply them in some form.

In (B) of Fig. 26 is shown a shell-type fly cutter of the same make, having two cutter blades. This cutter is made to fit on a standard shank which can be used for other cutters of various sizes. It can also be made so that it can be fastened to the spindle of the machine in a way similar to that described in the section on face mills.

Mountings for Milling Cutters. So that the milling cutter may function properly, it must be mounted on an arbor suitable for the job and to the cutter. This mounting must be as rigid as possible and, above all, must run true.

Milling Machine Spindles. Milling machine spindles are made to certain standards approved by the milling machine manufacturers' group of the National Tool Machine Builders' Association. In the construction of the spindle, its front, or nose, has been adapted to modern cutting speeds and forces. It possesses the accuracy and rigidity neces-

sary to assure good connections between the cutter head or the cutter and the arbor, and between the arbor and the machine spindle. Fig. 27 and Table IX give the essential dimensions for these spindles.

Milling Machine Arbors. Plain, side, slot-milling cutters, etc., are placed on a milling machine arbor which may or may not have a

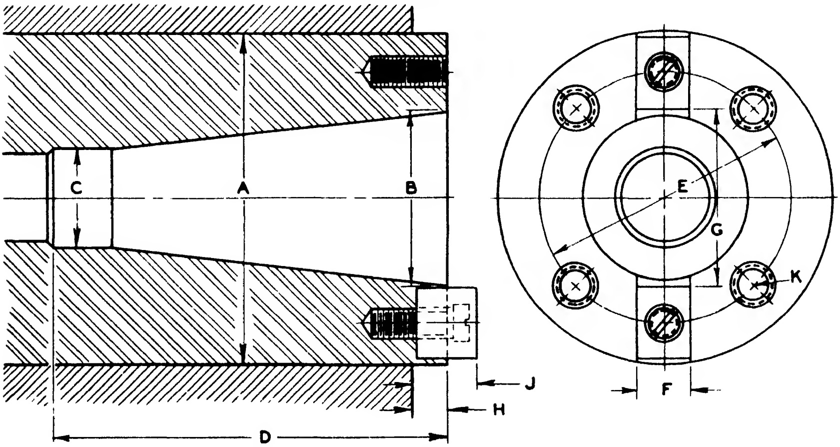


Fig. 27. The Standard Milling Machine Spindle

standardized spindle end. The cutters are keyed to the arbor and clamped between bushings. All parts of this connection, that is, the arbor, bushings, and the face of the clamping nut, should be ground so as to run true and at right angles to the axis of the arbor. If this is

TABLE IX. ESSENTIAL DIMENSIONS FOR STANDARD MILLING MACHINE SPINDLES

Taper No.	A	B	C	D	E	F	G	H	J	K
10		5/8	0.3785 0.3735	1 1/2						
20		7/8	0.504 0.498	2						
30	2.749 2.7488	1 1/4	0.692 0.685	2 7/8	2.130 2.120	0.6255 0.6252	1.315 1.285	5/16	1/2	3/8 x 3/4 #16
40	3.499 3.498	1 3/4	1.005 0.997	3 7/8	2.630 2.620	0.6255 0.6252	1.819 1.807	5/16	5/8	1/2 x 15/16 #13
50	5.0618 5.0613	2 3/4	1.568 1.559	5 1/2	4.005 3.995	1.0006 1.000	2.819 2.807	1/2	3/4	5/8 x 1 1/8 #11
60	8.7180 8.7175	4 1/4	2.381 2.371	8 5/8	7.005 6.995	1.000 0.999	4.819 4.807	1/2	1 1/2	3/4 x 1 3/8 #10

not done, there is the risk of bending the arbor when tightening the clamping nut. The cutters then would not run true. Since there is so wide a variety of arbors available from various manufacturers for various machines, no attempt will be made here to describe them or give their dimensions and specifications. In any case, arbors for carbide cutters at this stage of development are no different from arbors for other cutters. Any standard manufacturer's catalog will give available styles and sizes.

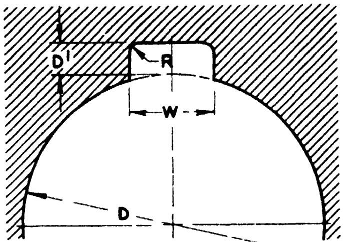


Fig. 28. Standard Keyway for Milling Cutters

Arbors are usually made of carbon tool steel, hardened and drawn to about 35-40 on the Rockwell C scale, and ground to run true. The purpose of hardening the arbor is to make it capable of resisting scratching by the cutters and bushings. While hardening increases the tensile strength of the material, it does not add to its rigidity. Although some tool engineers prefer to leave the arbors soft, most of them prefer hardened arbors. Arbors are made in diameters suitable for cutters of any particular size, and in lengths sufficient to accommodate a wide variety of work. Arbors are provided with one or two hardened sleeves for rigid support by the supporting arms of the machine.

Solid-shank, shell end mills and face mills should be mounted as short as possible to insure rigid supports. These are mounted on arbors which fit into the spindle as shown in Fig. 27. Again, a wide variety is available and will not be listed here. For face mills, arbors have a tongue in the middle section that fits into the slot of the face mill and acts as a driver. The mills are fastened to the spindle by means of four screws and are thus held securely in place.

Standard Holes and Keyways. Plain, side, and slot milling cutters are driven by a key in the arbor and, to a certain extent, by the frictional resistance between the bushings and the cutter hub. For heavier cuts, the frictional resistance alone would not be sufficient, hence, it is necessary to rely on the key for drive. Cutters of this type are made with holes and keyways as shown in Fig. 28. The keyway extends across the entire thickness of the cutter hub. Table X gives the limits to which the holes should be finished, the size of the keyway, and the radius of the fillet at the bottom corners of the keyway.

Since the key in the milling machine arbor prevents the cutter from slipping, the cutting forces on the cutter teeth exert a shearing force on the key. To resist this force, the key should have adequate length and width. How this works in actual practice may be illustrated by the following simple computation:

A slot milling cutter 6" in outside diameter, having a 2" hole and a

TABLE X. STANDARD HOLES AND KEYWAYS IN MILLING CUTTERS

Diameter of Hole			Size of Keyway						
D			W			D		R	
Nominal	Max.	Min.	Nominal	Max.	Min.	Max.	Min.	Radius	Hub Diameter
1/2	.501	.500	3/32	.106	.99	.0678	.0578	.020	
5/8	.626	.625	1/8	.137	.130	.0835	.0735	1/32	
3/4	.751	.750	1/8	.137	.130	.0825	.0725	1/32	
7/8	.876	.875	1/8	.137	.130	.0825	.0725	1/32	1/3/8
1	1.001	1.000	1/4	.262	.255	.114	.104	3/64	1 1/2
1 1/4	1.251	1.250	5/16	.325	.318	.145	.135	1/16	1 3/4
1 1/2	1.501	1.500	3/8	.410	.385	.176	.166	1/16	2 1/8
1 3/4	1.751	1.750	7/16	.473	.448	.208	.198	1/16	2 3/8
2	2.001	2.000	1/2	.535	.510	.208	.198	1/16	2 11/16

width of 1/2'', is to cut a slot 1/2'' wide and 1'' deep. The cutter has 8 teeth tipped with sintered carbide. The cut is to be made in medium hard steel at 250 r.p.m., with a chip load of .005''. Assume it is desired to learn the cutting speed, the hp necessary to make the cut, the force on the teeth and at the keyway, and the shearing stress in the key.

The cutting speed can be found by substituting 6 for D and 250 for N in the formula

$$S = \frac{3.14 \times D \times N}{12}$$

resulting in

$$S = \frac{3.14 \times 6 \times 250}{12} = 392.5 \text{ f.p.m.}$$

To find the horsepower, substitution is made in this formula,

$$hp = KdfNTw$$

resulting in

$$hp = 1.9 \times 1 \times .005 \times 250 \times 8 \times 0.5 = 9.5$$

The value for K is taken from Table IV in Chapter XIII.

To find the average force acting on the teeth, the hp and the cutting speed are substituted in the formula

$$P = \frac{hp \times 33{,}000}{S}$$

giving

$$R = \frac{9.5 \times 33{,}000}{392} = 798$$

This force acts at the radius of the cutter, which is 3''. However,

the keyway is at a radius of 1". The keyway must resist a force which is inversely proportional to the distance from the center of rotation. In this case, then, the force is three times greater. Therefore, the force on the key is 3 × 798 or 2,394 pounds.

This force tends to shear off the key at a plane along the diameter of the arbor and the hole in the cutter. The shearing force is resisted by the resistance of the key material to the shearing action. This is the product of the area of the key in the shear plane and the shearing strength of the material of which the key is made. Stating this in the form of a formula gives

$$F = LwS_S$$

in which

F = the force on the key in pounds
L = the length of the key in inches in the cutter hub
w = the thickness of the key in inches
S_S = the shearing stress of the key material in p.s.i.

Since it is desired to find the shearing stress, formula $F = LwS_S$ may be transposed, giving

$$S_S = \frac{F}{Lw}$$

When known values are substituted for letters in this formula, it is found that

$$S_S = \frac{2{,}394}{0.5 \times 0.5} = 9{,}600 \text{ p.s.i.}$$

The value obtained is not excessive, since the shear strength of materials ordinarily used for keys is: cold-drawn steel, 50,000 p.s.i.; machinery steel, 45,000 p.s.i.; casehardened machinery steel, 60,000 p.s.i.; and hardened and drawn steel, 90,000 p.s.i. As the stress in the problem is only 9,600 p.s.i., a cold-drawn steel could be used. This would give a ratio of

$$\frac{\text{shearing strength}}{\text{shearing strength}} = \frac{50{,}000}{9{,}600} = \frac{5.2}{1}$$

or a factor of safety of 5.2. This means that the key is about 5 times as safe as it needs to be if everything is in perfect condition at all times. If hardened steel were used for the key, the factor of safety would be

$$\frac{90{,}000}{9{,}600} \text{ or } 9.4.$$

The usual practice is to provide a factor of safety of from 5 to 10. This holds the shearing stress in the key to a minimum, providing for

TABLE XI. RIGIDITY RATIOS OF ARBORS

Diameter of Arbor in Inches	Rigidity Ratio
1	1.0
1 1/8	1.6
1 1/4	2.4
1 1/2	5
1 3/4	8.4
2	16
2 1/4	25.6
2 1/2	38
3	81

emergencies which may occur when the greatest care is exercised. Should the key be sheared off, slipping of the cutter will occur, of course. This would mean the stalling of the machine, which is usually disastrous to carbide-tipped teeth. Since there is also present the frictional force between the cutter and the bushing, the 5 to 10 safety factor is adequate for this type of drive.

When it becomes necessary to increase the resistance to the shearing force, the cutter may be designed with a greater width of hub, or a heat-treated alloy steel can be used for the key.

For narrow slot cutters, the hardened key should extend from the cutter on each side, entering the keyways in the bushings. This adds to the area in shear and thus to the resistance to shear by twice the thickness of the key multiplied by half of its depth.

Rigidity of Arbors. In the latest types of milling machines, designers have given much study to the subject of greater rigidity of the various parts of their machines. This is particularly true of those parts which are subjected to heavy strains and shock loads. Carbide-tipped cutters require rigid machines, arbors, and work holding fixtures. The efforts of machine designers too often are nullified because users mount the cutters on arbors that are too small in diameter with the result that the cutters are not rigid enough.

The rigidity of the arbor is not dependent on the strength of the material from which it is made, but rather on its diameter when a given length is considered. Although it is possible to increase the strength of a steel arbor by proper heat treatment, the process will not increase its rigidity. Rigidity varies directly as the fourth power of the diameter. This means that if a 1" arbor is assumed to have a rigidity ratio of one, a 2" arbor of the same length will have a rigidity ratio of 16. This is the reason for specifying milling cutters having holes as large as possible, and the use of arbors conforming to these holes. Table XI gives the ratios of rigidity of various sizes of arbor.

Design of Reamers. The purpose of reaming is to finish a

TABLE XII. REAMERS COMPARED BY NUMBER OF FLUTES

Diameter of Reamer in Inches	Number of Flutes	
	High-Speed Steel	Carbide Tipped
1/4 to 1/2 inclusive	6	4
17/32 to 1 1/4 "	8	6
1 9/32 to 1 3/4 "	10	6
1 25/32 to 2 1/4 "	12	8
2 9/32 to 2 3/4 "	14	8
2 25/32 to 3"	16	8

previously drilled or bored hole. The problem may be to finish the hole to predetermined exact tolerances, to produce a smooth finish on the walls of the hole, or both. In the design of reamers, it is essential to guard against any factor that tends to deviate from the above requirements. Chapter XII dealt in part with the use of carbide-tipped reamers and the methods of their employment to obtain the best results. A review of that part of Chapter XII will be helpful here.

Classification of Reamers. Reamers may be classified in many ways, according to their construction or use. One of the classifications commonly used is as follows:

1. Hand reamers
2. Expansion reamers
3. Taper shank reamers
4. Morse, and Brown & Sharpe taper reamers
5. Shell reamers
6. Expansion chucking reamers
7. Fluted chucking reamers
8. Rose chucking reamers
9. Stub screw machine reamers
10. Taper pin reamers
11. Taper bridge reamers
12. Taper car reamers
13. Taper pipe reamers
14. Adjustable reamers
15. Special reamers

All these reamers may have blades tipped with either cast or sintered carbides. However, expansion and small taper reamers tipped with carbide have not thus far been developed to any extent.

Number of Cutting Edges. The number of lands or cutting edges in hand or machine reamers is well standardized. Reamers used for chucking operations usually have fewer teeth than standard reamers. The same is true of adjustable reamers. Table XII gives the number of flutes for solid and shell-type reamers which may be tipped with cast carbides or made entirely of cast carbide.

Some machinists prefer an uneven number of teeth in a reamer. It

is claimed that this will prevent chattering, a common source of trouble in working with roughly finished holes. Other machinists prefer their reamers to have an even number of cutting edges but to have them unevenly spaced. This, too, prevents chattering. The disadvantages of an uneven number of teeth or of unevenly spaced flutes is the difficulty in measuring the diameter. In uneven spacing, the diameter of the flutes can be measured accurately if the diametrically opposed flutes are spaced exactly 180° apart.

Fig. 13 in Chapter XII illustrated a solid type of reamer which was tipped with carbide. In a reamer of this variety, the carbide need only extend over a short length of the tool, since this is sufficient for most problems arising out of machine reaming. For hand reamers, the blades should extend over the entire body of the reamer. The same is true for taper-type reamers.

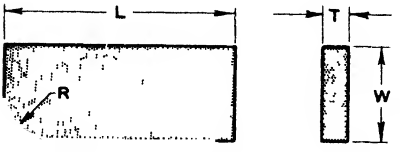


Fig. 29. Details of the Carbide Reamer Blade

The sort of blanks that are usually used for tipping machine reamers are shown in Fig. 29. The dimensions and suggested diameters for which they may be used are given in Table XIII.

Shell-Type Chucking Reamer. In Fig. 30 is illustrated a typical, solid-type shell reamer of the variety most often used for chucking work. This reamer has chamfered edges and is used for the removal of a considerable amount of material from the bored or previously drilled hole. These reamers are not intended to finish the work to extreme accuracy.

A special type of shell reamer tipped with sintered carbide is shown in Fig. 31. The feature of this reamer is its adjustability. When the outside diameter becomes smaller because of wear, adjustment can be made by loosening the screws visible in the illustration and forcing the blades forward the desired amount, thus enlarging the reamer. After adjustment, the reamer should be ground on the periphery and on the chamfer in order that all blades will be equidistant from the center.

TABLE XIII. SPECIFICATIONS FOR REAMER BLADES

Reamer Diameter	T	W	L	R
1/4 to 3/8	1/32	1/16	1/2	1/4
13/32 to 1/2	3/64	3/32	1/2	1/4
13/32 to 1/2	3/64	3/32	11/16	1/4
17/32 to 5/8	1/16	1/8	1/2	1/4
21/32 to 3/4	3/32	3/16	11/16	5/16
25/32 to 1	1/16	13/16	3/4	5/16
1 1/16 to 1 1/2	1/16	17/16	3/4	3/8
1 9/16 to 2 1/2	1/16	3/8	3/4	3/8

Reaming Speed and Feed. Reaming practices were discussed in Chapter XII. This consideration will be limited to the more specific information regarding reaming speeds and feeds, and the amount of material to be left for finish reaming.

Machine reamers are operated at about half the speed commonly designated for drills. Some authorities advocate a speed two-thirds that used for drills. If speeds for reamers are too low, their ability to produce is reduced at no gain in tool life. On the other hand, if the speed is too high, the reamer will wear prematurely, and may score the hole; that is, leave marks on the wall of the hole. When reaming hard material, care should be exercised that the speed not be too high. Too high a speed under such conditions can ruin a reamer quickly. Carbide-tipped reamers, however, are usually operated at the same speed as were the drills that preceded them on the job.

The feed of the reamer for general work should be about three times that recommended for the drill of the same size. Feeds are governed by the finish required. A coarse feed tends to produce revolution marks and rough holes. Too fine a feed, on the other hand, makes the reamer idle in the cut and causes it to wear too fast. For a given job a certain amount of experimenting will be necessary before the best feed for reaming is established. Generally, the feed for the reamer varies between .014'' and .040'' per revolution.

Reaming Allowances. The amount of material allowed for machine reaming is dependent upon the size of the reamer and the material to be reamed. Some mechanics insist on leaving 1/64'' total allowance for reaming regardless of the size of the hole to be reamed. This is obviously a poor practice for it throws an excessive burden on small size reamers and, in some cases, makes properly finished holes impossible. The allowances listed in Table XIV are recommended for long tool life and satisfactory finishes.

Selection of Reamers. Machine reamers are manufactured to

Fig. 30. Shell-Type, Carbide-Tipped Reamer

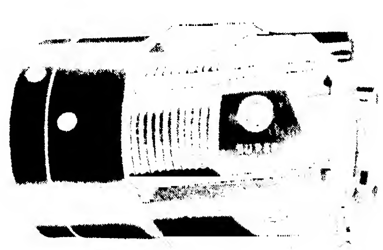


Fig. 31. Shell-Type Adjustable "Super-Reamer"
Courtesy of the McCrosky Tool Corp.

TABLE XIV. ALLOWANCES FOR REAMING

Diameter of Reamer in Inches	Amount of Material Removed Each Side in Inches
1/16 to 1/4	.003 to .005
13/32 to 1/2	.005 to .008
17/32 to 1	.007 to .010
1 1/32 to 2	.010 to .015

certain tolerances which are listed in Table XV. Every reamer will cut a hole larger than its own size. How much larger that cut will be depends on the material to be cut and the amount of material left for reaming. It has been found through research that a machine reamer will cut .0007'' over its own size when working on certain types of bronzes and cast iron. This is true for holes up to 1 1/2'' and, in some

TABLE XV. TOLERANCES OF REAMERS

Diameter of Reamer in Inches		Tolerances
1/16 to 1/4	inclusive	.0001 to .0004
17/64 to 1	''	.0001 to .0004
1 1/64 to 2	''	.0002 to .0006

cases, even larger. Therefore, in selecting a reamer for a given job or in designing one for a particular purpose, consideration must be given to the manufacturing tolerances to which the reamer is made, the amount it will run over its own size, and to the resultant tolerance.

For example, if a .500'' plus .002'' and minus .000'' hole is to be reamed, the diameter of the reamer at the start should not be so large that it would produce holes exceeding the tolerance. Since the maximum hole diameter may be .502'' and the 1/2'' reamer has a tolerance of .0001'' to .0004'', and since it will ream .0007'' over its own diameter, it is seen that the reamer must be at least .0008'' and possibly .0011'' smaller than the maximum dimensions of the hole. Allowing at least .0004'' for the margin of safety, one may use a reamer at least .0015'' (.0011 + .0004) under the .502'' diameter. The .5005'' diameter machine reamer will produce holes .5013'' when it is made to the minimum dimension, and .5016'' holes when it is made to the upper limits of reamer tolerance.

It will be seen from the foregoing that the .5005'' reamer will produce holes near the maximum size and will have considerable wear life before it begins to ream holes under .500''. The following formula, then, may be used for the selection of machine reamers which will give satisfactory performance:

$$D = d - (.0007 + t + T \times 20\%)$$

In which

D = the actual diameter of the reamer in inches
d = the maximum diameter of the hole desired, in inches
t = the maximum reamer tolerance in inches
T = the maximum product tolerance in inches

The application of this formula may easily be checked if it is assumed that known values in a reaming problem are d = .502, t = .004, and T = .002. These values are substituted in the formula, giving

$$D = .502 - (.0007 + .0004 + .002 \times .20) = .5013$$

Machine Reamers. An unevenly spaced machine reamer is shown in Fig. 32. It should be noted that this reamer has a margin which is the cylindrical land. This is usually from .003" to .005" wide. Also evident from the illustration is the land proper, the width of which

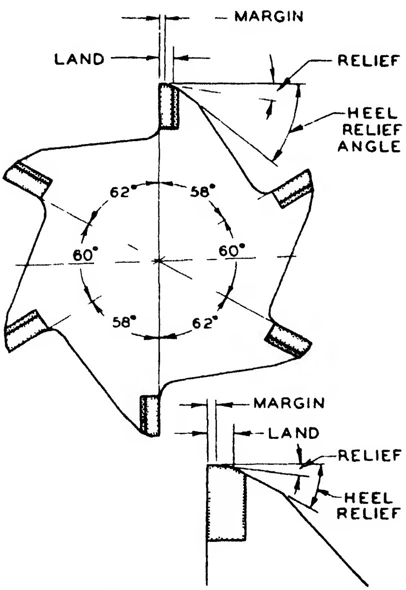


Fig. 32. Details of a Reamer Having Unevenly Spaced Blades

Fig. 33. A Typical, Counterbore Cutter. The Pilot for This Tool Slides Through the Hole in the Nose
Courtesy of the Eclipse Counterbore Co

is usually .025" to 1/32". The land has a relief of from 3° to 4°. Behind the land relief is the heel relief of 8° to 10°.

For reaming hard, abrasive material, the land is small and the

amount of margin practically zero. Hand reamers have the relief ground to the very edge to make cutting easier. All reamers should be tapered toward the back. The amount of tapering should be very small, usually from .0001'' to .0002'' per inch of length.

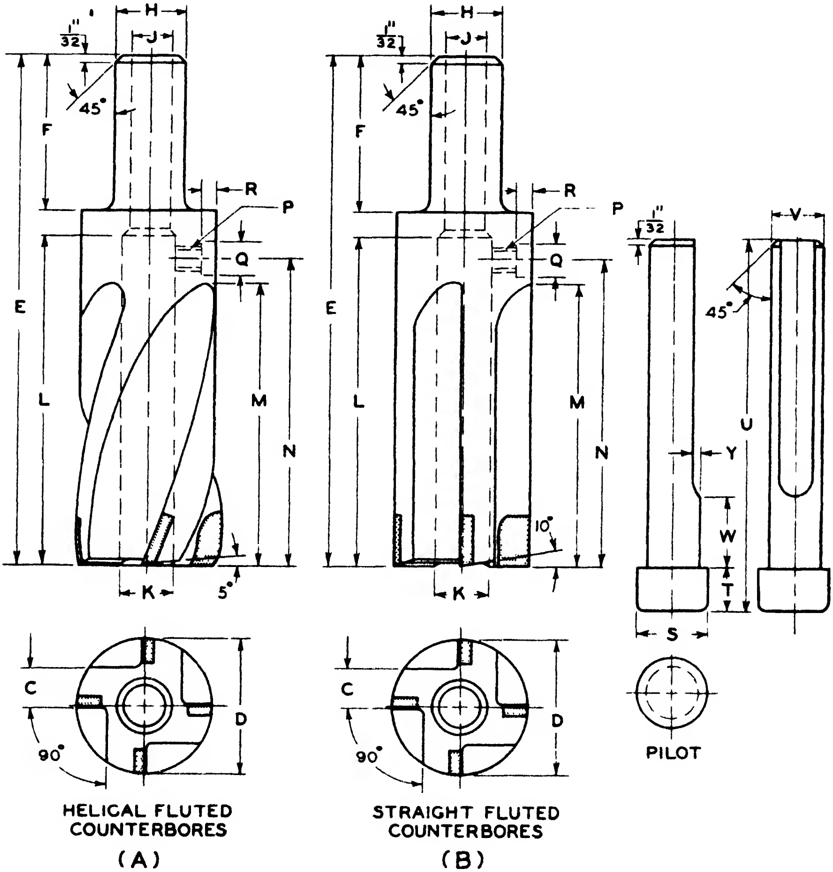


Fig. 34. Helical, Fluted Counterbore and Straight, Fluted Counterbore with Details for Dimensions

Design of Counterbores. Counterbores are multiple-edged cutting tools used mostly for the enlargement of a portion of a hole that has been previously drilled, reamed, or bored. If the counterboring action is of a shallow nature, as in squaring the surface around a hole is previously drilled to assure a flat seat for a bolt head or nut, the operation is known as spot-facing. When the enlargement of the hole is for the purpose of providing an undercut for a screw head, the operation is known as counterboring.

Counterbores are used extensively in the metal working industries, in tool shops, jobbing shops, and in production shops. It may be said also, that few tools are more generally abused in use or more carelessly maintained than counterbores. It is for this reason that designers and tool engineers should give particular attention to the features of their design.

As can be seen from Fig. 33, the counterboring tool consists of a body and a special shank. The body has a hole through it to receive the

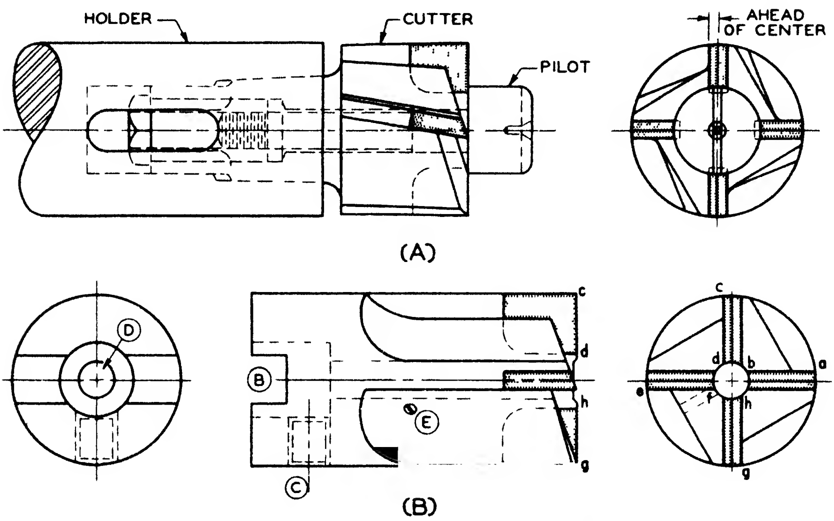


Fig. 35. The Interchangeable Types of Counterbore

required size shank. When used it is held in a special socket, adapted to the machine spindle.

Construction of Counterbores. Counterboring tools may be designed for cutting metals or nonmetallic materials. Figs. 34 and 35 show counterbores made of three parts: the body with the cutting flutes and edges, the pilot, and the setscrew locking the pilot.

The helical, fluted counterbore, shown in (A) of Fig. 34, permits rake angles on the blades. This results in easier cutting of the material with less consumption of power. However, since the counterbore with a straight flute is easier to make, it is preferred by many shop operators. The straight flute counterbore as shown in (B) of Fig. 34 can be used successfully on automatic screw machines and in turret lathes. Longer counterbores are used in drill presses and other machines.

Table XVI gives formulas for proportions as well as dimensions for the stub-type counterbore shown in Fig. 34. By changing their length, these tools can be used for operations on other machines as well as on screw machines.

TABLE XVI. FORMULAS FOR STUB-TYPE COUNTERBORES AND PILOTS

Counterbore

D = maximum diameter of counterbored hole
C = 1/4 D
E = 2D + 1 3/4
F = 1 to 1 1/4"
H = D for counterbores up to 5/16" diameter, 5/16" for counterbores from 5/16" to 1/2" diameter, 1/2" for counterbores 1/2" to 1" in diameter
J = 5/32" for counterbores from 7/32" to 1/2" diameter, 9/32" for counterbores .501" to 1" diameter
K = V + .001
L = 2 D + 1/2
M = 2 D + 1/8
N = 2 D + 9/32
P = .125 for counterbores up to 1/4", .141-40 for counterbores 1/4" to 1/2", .166-40 for counterbores 1/2" to 1"

Pilot

S = minimum diameter of drilled hole
T = S up to 5/16" diameter, 5/16" to 1/2" above 5/16" diameter
U = L + T − 1/16
V = S − .031 to .0625
W = 3/4 to 1 1/2 S
Y = 1/64 to 1/32

Interchangeable-Type Counterbores. Another type of counterbore and spot-facing tool is the variety shown in (A) of Fig. 35. These are usually made in three parts: the pilot, the cutter, and the shank. With this type of construction, the same shank can be used for a whole series of different sized cutters. Similarly, the cutter can take various sizes of pilots suited to the work to be done.

The teeth of the cutters are ground ahead of the center so that a shearing action on the material is produced. This is said to eliminate chatter and hogging. The teeth of the cutter can be made straight instead of helical when soft brass or similar material is to be machined.

The shell-type counterbore is shown in (B) of Fig. 35. This tool is preferred by many machinists because it is easy to store and maintain. The cutter of the tool consists of a body with four teeth, ab, cd, ef, and

gh; a pilot hole, D; and a slot, B. The tool is mounted on a shank which has a keylike projection fitting into the slot, B. The cutter is then locked to the shank by a setscrew, C. The pilot can be varied to suit the work,

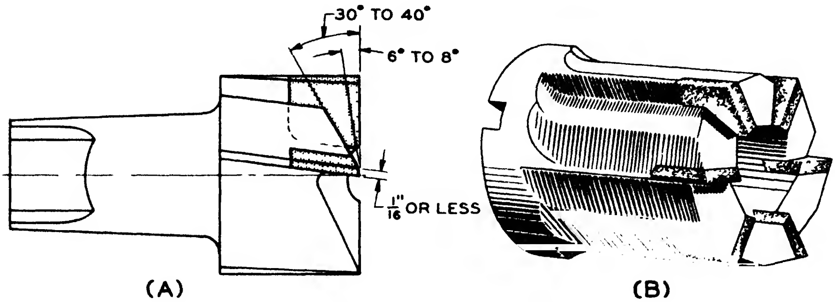


Fig. 36. The Shell-Type Counterbore

thus making this tool an interchangeable counterbore, and is locked in position by setscrew E.

In (A) of Fig 36 is shown another type of construction of the counterbore in which the tool fits into a shank and is driven by a tang at the end of its own shank. If the guiding of the counterbore driving shank is done by a steel bushing, as in any drill jig, this type of counterbore can be used without a pilot.

Fig. 37. Stub-Type Counterbore with Tapered Shank
Courtesy of the Eclipse Counterbore Co

Another counterbore with six carbide-tipped teeth is shown in (B) of Fig. 37. This tool is one of the larger variety since, generally speaking, counterbores are made with two or four teeth, a number usually suffi-

cient to meet all requirements imposed by varying jobs. In this construction, the counterbore has a fairly large hole running through it to receive a shank which, when sufficiently advanced, can also act as a pilot. The driving of the counterbore by the shank is done through a key that either fits in the shank or is made solid with it, and the slot of the counterbore.

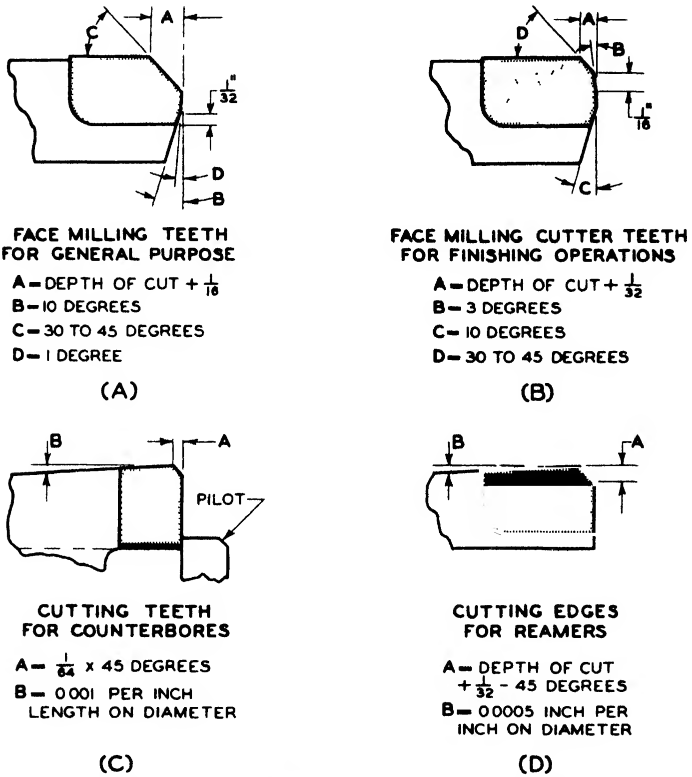


Fig. 38. Shapes of Carbide Teeth for Multiple-Pointed Cutting Tools

Combination Tools. For facing the bosses of the holes and chamfering their edges at the same time, a tool of the type shown in Fig. 37 is often used. The tool consists of the body with cutting edges tipped with carbide, and a pilot which goes through the body and the stub shank, being held in place by a nut. The chamfering operation is similar to countersinking. In this construction, some blades are assigned to the facing operation while others do the countersinking.

General Shape of Teeth. Shapes of the teeth commonly used for face milling were shown in Fig. 11. These shapes represent good practices and are recommended to any individual who is particular about

TABLE XVII. SPEEDS AND FEEDS FOR COUNTERBORES

Material Cut	Cutting Speeds in f.p.m.	Feeds per Revolution
Soft steel	250 to 300	.004 to .008
Alloy and tool steel	180 to 200	.004 to .008
Cast iron		
Soft	300 to 350	.006 to .012
Medium hard	200 to 250	.006 to .008
Brass	400 to 700	.006 to .012

cutter performance. For general shop work, however, the shapes of the teeth most often used are shown in Fig. 38. At (A) is the shape commonly used for general shop work—that is, for milling all sorts of metals. At (B) is shown the shape of the cutter teeth used for finishing operations on metals. At (C) and (D) are shown the general shapes of teeth for counterbores and machine reamers. It should be noted that the cutting edges have a back relief on the periphery so as to relieve the back portion of the counterbore or the reamer. This prevents their rubbing on the wall of the hole, and is considered good design practice. Aside from the back relief, however, these tools should be held in holders in such a way that they are in perfect alignment with the hole in the work. If this is not done, poorly finished holes will result and the tool life will be shortened.

Counterbore Speeds and Feeds. Counterbores cut inside the material, therefore, they are confined in the material and for this reason are subjected to greater heating action than is found in milling cutters. For this reason, they should be operated at speeds somewhat lower than those specified for milling cutters.

Table XVII gives suggested speeds and feeds for counterbores. The values given in the table can be used as a guide, and may be increased or decreased to suit the conditions of the job. When the cut is wide, the speed should be decreased by about 25%. For the dry cutting of steel, the speed should be decreased by about 10 to 15%. If a coolant is used, it should be played on the tool in copious quantities so as to keep the tool cool at all times.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What is the outstanding characteristic of milling operations performed with sintered carbide-tipped cutters?

2. Why are carbide-tipped cutters designed with fewer teeth than high-speed steel cutters?

3. Why should the r.p.m. of a carbide-tipped cutter exceed 100?

4. What is the name of a milling cutter having only one tooth?

5. Explain the meaning of the term, "reamer."

6. What governs the amount of material that should be removed with a machine reamer?

7. At what speeds are machine reamers operated?

8. How many teeth should a 2" reamer have?

9. Define a counterbore.

10. Name the main parts of a counterbore.

11. At what speeds can carbide-tipped counterbores operate?

12. Why is it desirable to have back taper on counterbores?

13. What factors determine the number of teeth in a carbide-tipped milling cutter?

14. What should be the factor of safety of the arbor key in operation of the milling-machine cutter?

15. What should be the number of teeth in a 12" milling cutter taking a cut 1/8" deep, 8" wide, with a feed per tooth of .006", at a cutting speed of 400 f.p.m. in a 15 hp machine when the value of factor K is 2?

16. How many teeth should a 12" carbide-tipped milling cutter have according to general shop formula?

17. What is the rigidity ratio of a 2" diameter milling machine arbor as compared with a 1" arbor?

18. A carbide-tipped machine reamer is to ream a .750" plus .002" and minus .000" hole. The work is done in soft steel. What should be the exact diameter of the reamer for this job?

REFERENCES

Cooper-Bessemer Co. "Meehanite Milling Cutters, Carbide Tipped," Bulletin 100-TC.

Eclipse Counterbore Co. "High Production Tools," Catalog No. 35.

Ex-Cell-0 Corp. Catalog M-338.

Field, M., and Bulloch, W. E. "Milling Cast Iron with Carbide Cutters," Machinery, Vol. 52, No. 1, Sept., 1945.

Ingersol Milling Machine Co. Engineering Specification Sheets.

Kronenberg, Dr. M. "Initial Contact of Cutter and Workpiece," Machinery, Vol. 52, Oct., 1945.

Lovejoy Tool Co., Inc. "Lovejoy Metal Cutting Tools for Modern Requirements," Catalog No. 27.

McCrosky Tool Corp. "Jack Lock Serrated Blade Milling Cutters," Bulletin No. 15-M.

Super Tool Co. Carbide Tipped Standard Milling Cutters, July, 1944.

Weddell Tools, Inc. Tri-Bit Cutters with Adjustable Blades.

CHAPTER XV

Selection of a Carbide

Difficulties in Making Selection. One of the tool engineer's and user's principal difficulty in the use of carbides often is the choice of the proper grade for the job at hand. What makes the problem so difficult is the fact that there are a dozen or more manufacturers of this material in the business and each of them designates his grades and types in a different manner. Not only is there no uniformity in designation among the various producers of carbides, but there is also no uniformity, grade for grade, in the products of each manufacturer. Nevertheless, an attempt will be made here to present a workable method by which one can select the type of carbide best suited for the job at hand.

Each manufacturer not only has his own trade-marked type and grade designations, but secrets in the making of the carbide itself. Entirely aside from the many restrictions that for years were placed on information about processes, it is almost impossible to learn from manufacturers exactly how each grade is made or even what is intentionally put into it! However, it can be stated in general that there are four principal types of carbide cutting materials. The first of these is straight tungsten carbide with whatever impurities there may be in the material as the result of the manufacturer's inability to refine adequately the raw ore or purchased metals. Secondly, there is the tungsten-tantalum mixtures which may, in some cases, contain slight amounts of titanium. Third, there are the tungsten-titanium carbides which may also contain small amounts of tantalum. The fourth and last type is what is usually called the triple carbides—tungsten-titanium-tantalum carbide alloys or mixtures which, depending upon the manufacturer, will have varying amounts of each ingredient. Columbium (niobium) may be, and often is, substituted for part or all of the tantalum. The two are such closely related elements that it is exceedingly difficult to separate them commercially.

There is one point, however, which should be stressed here: The wrong grade, if properly used, will be much more effective than the right one improperly used. For this reason, it may be well to summarize here, some of the points that have been mentioned in previous chapters. This will serve to emphasize the important factors of good operation which must be considered at all times. These points hold good not only in the use of carbides on machines especially built for them, but also in the conversion of older machines to the use of carbides. When real

TABLE I. MAKE AND GRADE OF CARBIDES PARALLELED BY TYPE OF WORK

Type of Work	Carboloy	Vascoloy-Ramet	Kenna-metal	Firthite
General machining of cast iron, nonferrous metals, and nonmetals	44A	2A68	K6	H or HB
Machining to close production limits, also, roughing-finishing and one-cut finishing on metals and nonmetals as above	883 or 905	2A5 2A8	K6	HA or HD
Machining steel (general purpose)	78B	EM	KM	TA
Light roughing and finishing of steel	78	E	K3H or K4H	T-16
Heavy-duty steel machining	78C	EE	KM or K2S	T-04 or T-89
Heaviest cuts (hogging) with slow speeds and heavy feeds (steel, cast steel) and intermittent cuts	77A	EE	KM or K2S	T-04

trouble develops in the shop or on the production line, it is possible in most instances to trace it to some other factor than the choice of grade or make of carbide.

Included in Table I in this chapter are some of the carbides produced by four of the principal manufacturers. This table will give a cross reference for purposes of comparison. It will be found in many cases that the grades, as shown across the column, are not parallel in all respects. The manufacturers themselves do not agree as to which of their grades approximate those of the others. They are paralleled here because their chief characteristics are alike, and because some sort of standard must be set up somewhere by someone.

In Table II are given the characteristics which any carbide should have in order to do the work listed. Part of a manufacturer's chart is used in Table III, showing the grades of carbide recommended for various materials and conditions of cutting. The use of this chart is not intended to confer any special endorsement of this particular manufacturer's products. Nevertheless, the company was one of the pioneers in the field and produces a complete and representative line of sintered

TABLE II. CHARACTERISTICS REQUIRED OF CARBIDES

Material	Job	Job Characteristics	Qualities Required in Carbides
Steels	rough milling, turning, boring, etc.	all types heavier feeds or cuts	cratering resistance toughness wear resistance edge strength
	finish turning, milling, boring, etc.	lighter feeds or cuts	same as above; even more abrasion resistance and edge strength
Plain cast and malleable irons	roughing	rough castings	toughness, wear resistance
	finishing	smoother castings, finishing cuts	abrasion resistance toughness
Steel types of cast and malleable irons, or irons with hard sections	rough and finish turning, boring, milling, etc.	all	cratering resistance abrasion resistance toughness
Aluminum alloys	rough and finish milling, turning, boring	general cutting	abrasion resistance toughness wear resistance ability to take and retain a keen edge
		form cutting	same as above; must also resist formation of built-up edge
Magnesium Zinc alloys Brass Bronze Plastics Fiber	turning, milling, boring, etc.	all	abrasion resistance wear resistance toughness

TABLE III. STANDARD CARBOLOY GRADES FOR VARIOUS MATERIALS AND SPEEDS

Material to Be Machined	S.A.E. Hardness	Recommended Carbide Grade for Specific Conditions of Feed, Speed, and Cut Depth					
		Light Duty (Cut 1/64 - 3/16, Feed .008-.012)		Medium Duty (Cut 3/16 - 3/8, Feed .015-.025)		Heavy Duty (Cut 3/8 - 5/8, Feed .025-.035)	
		f.p.m.	Grade	f.p.m.	Grade	f.p.m.	Grade
Steel							
Carbon	1010-1025	375-625 500	78	200-400 300	78B	100-200 150	78C
	1030-1095	300-500 400	78	150-350 250	78B	75-150 110	78C
Free-Cutting	1112-1120	375-625 500	78	200-400 300	78B	100-200 150	78C
	X1314-X1340	350-550 450	78	175-375 275	78B	90-160 125	78C
Manganese	T1130-T1350	250-450 350	78	125-300 200	78B	60-120 90	78C
Nickel	2015-2320	350-550 350	78	175-375 275	78B	90-160 125	78C
	2330-2515	300-500 400	78	150-350 250	78B	75-150 110	78C
Nickel-Chromium	3115-3140	300-500 400	78	150-350 250	78B	75-150 110	78C
	3145-3450	250-450 350	78	125-300 200	78B	60-120 90	78C

Molybdenum..........................	4130-4820	250-450 350	78	125-300 200	78B	60-120 90	78C
Chromium..........................	5120-52100	350-450 350	78	125-300 200	78B	60-120 90	78C
Chromium-Vanadium..............	6115-6195	250-450 350	78	125-300 200	78B	60-120 90	78C
		300-500 400	78	150-350 250	78B	75-150 110	78C
Cast Iron (No Alloy)..........................	hard	200-300 250	883	150-250 200	44A	75-125 100	44A
	medium	225-325 275	883	175-275 225	44A	90-140 115	44A
	soft	250-350 300	883	200-300 250	44A	115-165 140	44A
(Alloy)..............................	hard	225	883	175	44A	60-110 85	44A
	medium	250	883	200	44A	75-125 100	44A
	soft	275	883	225	44A	90-140 115	44A
Up to 25% Semisteel..............	....	275	883	225	44A	90-140 115	44A

Continued on owing page)

TABLE III (Continued)

Material to Be Machined	S.A.E. Hardness	Recommended Carbide Grade for Specific Conditions of Feed, Speed, and Cut Depth					
		Light Duty (Cut 1/64 - 3/16, Feed .008-.012)		Medium Duty (Cut 3/16 - 3/8, Feed .015-.025)		Heavy Duty (Cut 3/8 - 5/8, Feed .025-.035)	
		f.p.m.	Grade	f.p.m.	Grade	f.p.m.	Grade
Cast Iron (continued)							
Over 25% Semisteel..................	....	225	883	175	44A	75-125 100	44A
Chilled Rolls...........................	....	15-50 20	883	10-45 15	44A		
Malleable Iron...........................	hard	225	883	175	44A	60-110 85	44A
	medium	225	883	175	44A	60-110 85	44A
	soft	275	883	225	44A	90-140 115	44A
Brass and Bronze......................	hard	200-400 300	883	125-275 200	44A	75-125 100	44A
	soft	300-500 400	883	175-325 250	44A	100-150 125	44A

Castings	....	300-1000 450	883	200-500 300	44A	100-150 125	44A
Aluminum Bar Stock	....	300-1000 400	883	200-500 275	44A	100-150 125	44A
Zinc Alloy Die Castings	....	300-600 450	883	200-500 300	44A	....	......
Rubber	hard	350-600 475	883	250-450 350	883	....	......
	soft	500-1000 650	883	300-600 450	883	....	......
Copper	....	300-600 450	883	150-350 250	44A	....	......
Fiber	....	300-800 500	883	200-400 300	44A	....	......
Plastics	....	400-1000 600	883	250-500 350	44A	....	......

The top figures under the f.p.m. column denote the average range. The bottom figure gives a safe speed from which to start and work in either direction.

carbide cutting material. Whenever a Carboloy grade is shown, it should be understood that the user may equally well decide to buy or use a similar grade or type from another line.

By the same token, it should not be inferred from Table I that the four manufacturers mentioned there are the only ones in the field or even that theirs is the best grade of carbide produced. However, their material is widely used and is well distributed.

It will be seen from these tables that, in general, some half dozen grades of carbide will be enough to cover, if not the entire range of metal cutting operations, at least a very large part of them. Many companies produce more than five grades of carbide for cutting tools. Some, on the other hand, recommend only three grades. Some manufacturers have nine; others a dozen or more. It is obviously impossible to set up any sort of standard to cover all possible contingencies. Nevertheless, by making use of the simple method of grade selection given here, the operator, the tool engineer, the tool designer, and the purchasing department will be able to meet most situations involving the proper choice of a carbide.

Importance of Grinding. First among the factors that bring about successful machining with carbides probably is the grinding of the tools themselves. If milling cutters, for instance, are not properly sharpened, neither the best grade of carbide nor the grade best suited for the work at hand will do a successful and efficient job. One of the points which should always be stressed in the grinding room involves the removal of too much carbide in one pass. In general, no more than .002'' should be removed in the rough grinding operation. For finishing, not more than .0005'' should be removed in one pass during grinding. This does not mean that not enough carbide should be removed from the worn tool to get down to good, solid metal. Rather, if the removal of quite a bit of metal is necessary, it should be done by taking more than one pass with the wheel.

A quick answer to the problem of how much carbide metal should be taken from the tool can be obtained by a study of the tool surface through a magnifying glass having a power of at least 20 magnifications. If sufficient stock has been removed, fine, hairlike lines will be seen on the surface being ground. These will cross the cutting edges at several points. In general, it may be said that the finer the lines, and the fewer, the better will be the grinding job. The reason for this statement is that these hairlike lines are usually the starting points from which tool breakdown spreads.

Complete details on grinding operations for various types of tools have been given in previous chapters, but it may be well at this point to stress again, the following point. If diamond wheels are used for sharpening cutters, the wheel should not be allowed to touch the metal of the tool shank or the cutter body. The recommended practice is to allow the carbide inserts to project beyond the edges of the tool shank by about 1/32''. The reason for this is that steel loads up and ruins a

diamond wheel. If the wheel has been used on steel to any extent, it will overheat and will cause a checked surface when used on carbide. Even the best grade of carbide will be quickly ruined by the formation of such heat checks.

Importance of Design. Second among the most important factors to be watched in carbide milling operations is that of cutter design.

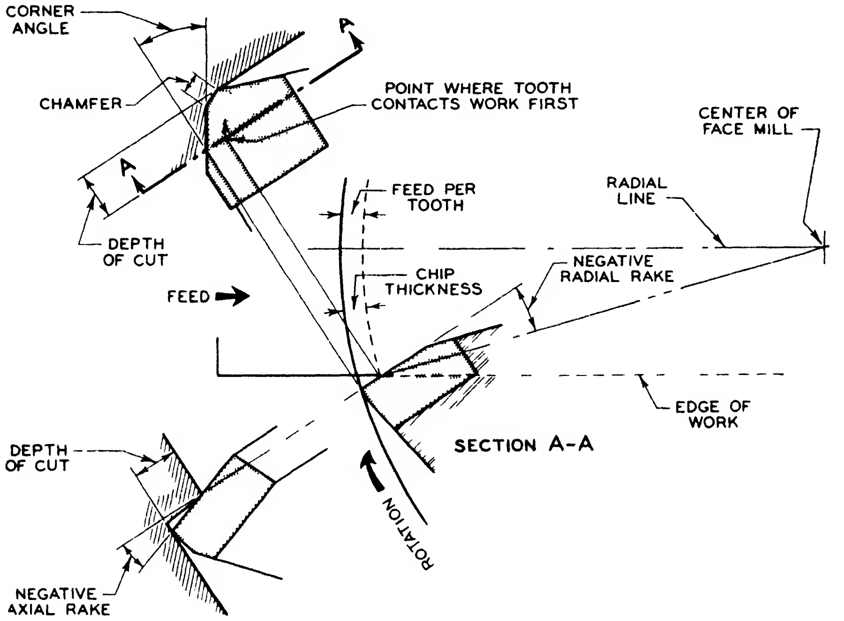


Fig 1. Impact Should Come Away from the Face of the Cutter at a Distance Equal to the Depth of Cut

The most perfect carbide and the finest job of grinding obtainable will not offset incorrect design of cutters. Most operators today are using both negative axial and radial rake in their operations, particularly if high speed or high finish is demanded. The general practice in both instances is to require 10° negative rake for the harder steels. This angle will, however, range all the way up to 5° positive for the softer steels and perhaps for some of the lighter and softer metals and materials. Sufficient information has been given in previous chapters to enable an engineer or operator to determine whether negative rake is the answer to his particular problem.

Another point regarding cutter design is that sufficient chip room must be provided so that the chip will not lodge in the chip space. Many edge failures with carbide tips are the result of sticky chips being pulled through the cut a second time. Such failures are usually not the fault of either the grinding operation or the carbide selection, but are

due solely to the carelessness of the engineer or tool designer who failed to allow sufficient chip space in designing the cutter.

The setup of the machine is another point which at all times must be regarded as a principal factor in successful machining with carbide. If the engineer does not take into consideration the particular demands of the job on which the cutter is to be used, again, neither the grinding nor the carbide grade selection can be blamed. One of the principal uses of negative rake angles is, of course, to keep the chip load away from the cutting edge and nose of the tool. A few simple rules can be repeated at this point which should help materially in determining the proper setup.

First, it should be remembered that the axial rake angle must always be sufficiently great to take the impact load away from the face of the cutter a distance equal to the depth of cut. This is shown graphically in Fig. 1. Also, the impact load should always be taken at a distance away from the outside edge of the tooth face at least equal to the thickness of the chip. It is quite possible to design and grind the tool with negative angles on the cutter and still, as the result of improper setup, have the load come squarely on the cutting edge or nose of the tool as it enters the cut. The only remedy for this is to make an exact check of the cutter setup at the time the tool is placed in the machine, thus making sure that the actual negative angle between the cutter and the work is what the tool engineer has specified.

It is obvious, of course, that by moving the work at right angles to the direction of feed, the impact load may be moved away from the outside cutting edge of the tool a distance equal to the chip thickness. For this reason, it is usually desirable, particularly in fast milling, to design the cutter with a bevel or corner angle so that the first impact load will be taken at some distance from the nose, or chamfer where one is used, especially when the cutter tooth has to enter the work with a large radius or draft angle on the corner. In some cases this corner angle has been known to go as high as 15°, 30°, and even 45°

Speeds and Feeds. Speeds and feeds, as has previously been pointed out, constitute one of the longest and most difficult subjects to understand and put to practical use of all the problems involved in cutting with carbide tools. It is understood, of course, that the selection of the proper grade of carbide has a great deal to do with the determination of speeds and feeds. Yet, as emphasized elsewhere in this book, even those who have been using carbides the longest have been unable to arrive at any agreement on the upper limits of feeds and speeds possible. There is one point, however, which should be emphasized at this time, although it has been stressed repeatedly before. When using carbides, there must be a heavy feed per tooth to avoid building up the chip load near the cutting edge.

At the present, most operators use a feed per tooth of between .004'' and .008''. Cutter length will certainly be lengthened, particularly in using negative rake angles, if the cutting edge reaches the cut with a thick chip. This chip thickness reaches its greatest point, which is equal

to the feed per tooth, when the edge of the work at the point of cutter entry is at the center line of the cutter and is parallel with the direction of feed. As pointed out before, if the work is moved in either direction at right angles to the feed, the chip load will change.

Another point which perhaps should be stressed in connection with speeds and feeds is that the carbide grade selected should usually vary with the hardness of the material being cut. This has been discussed at length in previous sections. To avoid extremes, it is well to keep in mind that a steel of 110 Brinell hardness should be milled at a speed of about 750 f.p.m. while heat-treated alloys with hardnesses up to 400 Brinell will mill best at a speed of about half that. If the speed is too high, the wear on the carbide increases to the point where the efficiency obtained by the use of the carbide is cancelled. If the speed is too low, a built-up edge results, bringing about cratering, short cutter life, and poor finish on the work surface.

Rigidity Necessary. As previously stressed, rigidity is one of the principal factors in any successful milling operations involving the use of carbides. Flywheels should always be used on milling machines, particularly if the cutter is small and/or the speed is low. A flywheel will dampen out vibration and chatter which may result from changes in load as the separate teeth enter and leave the cut. No matter how carefully the grinding may be done to insure uniform chip load on each tooth, there is always likely to be some slight variation.

The flywheel factor is particularly important in the use of carbides because sintered carbide cutters, in most operations, have fewer teeth than, for example, high-speed-steel cutters, or even those made of the cast carbides. Vibration and chatter are much more injurious to any grade of carbide than to any of the other cutting materials. If there is very much of either present, the damage to the tool will again become so great as to reduce or cancel the efficiency resulting from the use of carbide.

It will be seen, of course, that the higher the cutting speed, the less important is the need for this damping action, and, therefore, the less weight is required for the flywheel because the cutter body and the machine spindle provide enough flywheel effect. Conversely, the lower the cutting speed, the heavier should be the flywheel. A good, rule-of-thumb formula, perhaps, is to make the flywheels twice as large as the cutter and also larger than the largest gear on the main spindle. Flywheels weighing from 50 to 100 pounds are frequently used with good results. Another way to achieve much the same purpose is to specify and use a cutter with a heavy body. Such a cutter will, in itself, act as a flywheel.

The tables given to show the proper carbide grades have been greatly simplified. If the operator should find that starting with the suggested grade does not produce satisfactory results, and if he is certain that all the aforementioned factors governing milling setup practices have been fulfilled, he should then shift to another grade and or make of carbide.

Four Kinds of Carbide. It will be noted in Table II that steel cutting has been divided into three classifications as regards job characteristics. It should also be noted that the recommendations are entirely relative. That is, any grade of carbide will have more wear and abrasion resistance than any other corresponding class of cutting material. Some points about the material itself, however, might well be given further consideration. For example, the simple tungsten carbides are more wear resistant than the tungsten-titanium and tungsten-tantalum carbides, while the so-called triple carbides are in general tougher than the straight tungsten carbides. In the rough milling of steels where the chip loading is extremely heavy, a resistance to cratering is a first requirement. This calls for toughness, wear resistance, and edge strength under heavy impact loading. In finish milling, on the other hand, where the feeds and cuts are likely to be much lighter, a straight tungsten carbide is usually the best. The reason for this is that the cutting edge has to be more wear resistant under these conditions than where the heavier cuts or feeds are used. In cases where not only a heavy cut but also a finish cut must be made in one pass, the user may find that either of the two types of carbide will be successful, depending upon whether the job comes closer to rough milling or finish milling.

Cast and malleable irons have been classified into two groups in the chart in Tables II and III. This is because many of the present-day irons approach steel in their characteristics. In these cases it will usually be found that steel-cutting grades will give better results than the straight, iron-cutting grades. In the group of irons that are more like steel in their qualities should be included all those, such as armor plate, which have hard spots or hard sections spotted through them. Such sections are often the deciding factor in the selection of a carbide grade, since they call for toughness rather than wear resistance in a tool.

In cutting the simpler cast and malleable irons, however, abrasion resistance is likely to be the quality desired. For this reason, straight tungsten carbides are usually considered best. Because of the peculiar quality of the grain structure of tungsten and its carbides, it will be found best in most cases to use the more coarsely grained tungsten carbides, especially if the working sections are unusually rough. The tungsten carbides assure a greater toughness since tungsten, unlike most other metals, is exceedingly tough in its across-the-crystal resistance. This is due to the fact that most metals fracture across the crystal rather than in the boundary phase. Tungsten does not. It fractures at the boundary, leaving whole crystals of the metal intact. On smoother castings as well as on finished cuts, however, the operator will usually find that the finer grades of straight tungsten carbide will give longer tool life and a better finish.

Cutting the Light Metals. Unless for some reason a very heavy cut at an extreme speed has been decided upon, it will be found that the fine-grained, straight tungsten carbides work the best for most machining of aluminum, magnesium, and the light metal alloys. The reason

is that these carbides have the ability to take and retain a fine and keen edge, which is of particular importance in the machining, for example, of aluminum. At the same time, they provide a high degree of resistance to the abrasion that results from the aluminum oxides and other abrasive materials which are often a part of the chemical make-up of the alloy itself. However, in the form milling of aluminum, it may be found that trouble will arise as a result of the built-up edge on the tool. In this case, a grade of carbide containing small amounts of tantalum has proved of great advantage.

On the other nonferrous metals and on the nonmetallic materials such as plastics, rubber, etc., a fine-grained, straight tungsten carbide will usually be found to be the best material to use, or at least try as a starting point. In the machining of these materials, the important consideration is the ability of the tool to resist abrasion and wear, plus the ability to take and keep a keen edge. Because of this, the fine-grained carbide is usually the better choice.

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points of the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. How many principal types of carbide are there for use in cutting tools?
2. What are they?
3. How do they differ on the job?
4. What carbide should be specified for the rough milling of steel where the chip load is exceedingly heavy?
5. What should be specified for use in milling ferrous castings where a rough and finish cut must be made in one pass?
6. What should be specified for the cutting of magnesium?
7. Suppose the magnesium were to be milled at high speed with a very heavy cut?
8. What carbide would be recommended for the machining of a glass-reinforced plastic?
9. What is the usual effect of nickel when it is present as an impurity in a carbide?
10. What is the effect of niobium in the carbide mixture?
11. Would it be better to use a triple carbide or a W·Ti·C grade for machining aluminum on an old machine which has loose spindle bearings?
12. Aside from the choice of grade, what is the first factor in importance regarding the use of carbide? The second factor?
13. If quite a few hair lines appear on the carbide tip when it is examined with a magnifying glass after grinding, what should be done?

REFERENCES

Crump, Harry. "Machining Glass-Reinforced Plastic with Cemented Carbide Tools," American Machinist, March 29, 1945, Pp. 91-93.

Frommelt, H. A. "The Art of Metal Cutting," Machine Tool Blue Book, Vol. 40, No. 10 through Vol. 41, No. 6, October, 1944 to December, 1945.

Lucht, Fred W. "Selecting Carbides for Milling," Steel, January 8, 1945, Pp. 84, 85, 126.

CHAPTER XVI

Tool Materials, Assembly

Early Development of Cast Alloys. Elwood Haynes, one of the pioneers in the automobile business, is generally credited throughout the metallurgical industry with being the father of the cast carbides, or, as they are technically known, the cobalt-chromium-tungsten system of alloys. It was Haynes' work, begun in 1899, that brought to the attention of the machine industries the many remarkable qualities of these alloys. Haynes actually made some cast alloys in 1909 or 1910 and had it tested as a cutting material in the machining of cast iron. His cobalt-chromium-tungsten cutting tools were placed on the market in 1913.

Some of the more technically inclined may object to the use of the term "cast alloys" in connection with these cutting materials. In the earlier days of their use, reference was usually made to them as the "cast alloys," a term which was thought to be misleading and confusing because there are literally thousands of alloys that can be and regularly are cast into various desired forms. A few engineers insisted on calling these materials "cast cutting alloys." This was not a good choice because there are dozens of alloys of iron alone which are commonly used as cutting tools, many of them containing varying amounts of chromium, tungsten, or both. So, perhaps for want of a better word, "cast alloys" is the term that has come into rather general use in recent years, particularly since the introduction of the sintered carbides, which were described in the foregoing chapters of this book.

The term "cast alloys" is used for several reasons. The first is that except for the presence of chromium, the cast alloys, chemically, are very closely related to the sintered carbides in their composition. Secondly, because in every case with which the authors are familiar, the tungsten and usually the chromium are present in the form of a carbide. Some of the mixtures also include certain percentages of tantalum carbide. The cobalt, as in the case of the sintered carbides, is used primarily as a binder or matrix although it also appears to confer certain desired physio-chemical properties, particularly that of "red hardness," to both products.

Thus, the resulting alloy is chemically a carbide, even though it is prepared in a vastly different manner than the sintered carbides, and the end product may not have all the same, or in some respects, even similar qualities. The sintered carbides, as was pointed out earlier,

TABLE I. R.P.M. OF CUTTERS HAVING VARIED DIAMETERS AND OPERATED AT VARIOUS SPEEDS

Cutter Diameter in Inches	Revolutions per Minute Depending upon Cutter Speed in Surface Feet per Minute																			
	High Speed Steel						Cast Alloys						Tungsten Carbide							
	40	50	60	70	80	90	100	110	120	140	160	180	200	250	300	350	400	450	500	600
1/4	611	764	917	1070	1222	1375	1528	1681	1833	2140	2444	2750	3056	3820	4584	5348	6112	6882	7640	9168
3/8	408	509	611	713	815	916	1018	1120	1222	1426	1630	1832	2037	2546	3056	3563	4074	4584	5092	6112
1/2	306	382	458	535	611	688	764	840	916	1070	1222	1376	1528	1910	2292	2674	3056	3438	3820	4584
5/8	244	306	367	428	489	550	611	672	733	856	978	1100	1222	1530	1833	2139	2445	2750	3056	3666
3/4	204	254	306	357	407	458	509	560	611	714	814	916	1018	1273	1528	1783	2037	2292	2546	3056
7/8	175	218	262	306	349	393	436	480	523	612	698	786	872	1090	1310	1528	1746	1964	2183	2620
1	153	191	229	267	306	344	382	420	458	534	612	688	764	955	1146	1337	1528	1719	1910	2292
1 1/8	136	170	204	238	272	305	339	374	407	476	544	610	678	850	1019	1188	1358	1528	1698	2038
1 1/4	122	153	183	214	244	275	305	336	366	428	488	550	611	764	917	1070	1222	1375	1528	1834
1 3/8	111	139	166	194	222	249	277	305	332	388	444	498	554	695	833	972	1111	1250	1389	1666
1 1/2	102	127	153	178	204	229	254	280	305	356	408	458	509	637	764	891	1019	1146	1273	1528
1 3/4	87	109	131	153	175	196	218	240	262	306	350	392	436	545	655	764	873	982	1091	1310
2	76	95	114	133	153	172	191	210	229	266	306	344	382	477	573	668	764	859	955	1146
2 1/4	68	85	102	119	136	153	170	187	204	238	272	306	340	425	509	594	679	764	849	1018
2 1/2	61	76	92	107	122	137	153	168	183	214	244	274	305	382	458	535	611	688	764	916
2 3/4	55	69	83	97	111	125	139	152	166	194	222	250	278	345	417	486	556	625	690	834

3	51	64	76	89	102	115	127	140	153	178	204	230	255	318	382	446	509	573	637	764
3 1/4	47	59	70	82	94	106	117	129	141	164	188	212	234	295	353	411	470	529	590	706
3 1/2	44	54	65	76	87	98	109	120	131	152	174	196	218	270	327	382	437	491	540	654
3 3/4	41	51	61	71	81	92	102	112	122	142	162	184	204	255	306	357	407	458	509	612
4	38	48	57	67	76	86	95	105	114	134	152	172	191	239	286	334	382	430	477	572
4 1/2	34	42	51	59	68	76	85	93	102	118	136	152	170	210	270	297	340	382	420	540
5	30	38	46	53	61	69	76	84	92	106	122	138	153	191	229	267	306	344	382	458
5 1/2	28	35	42	49	55	62	69	76	83	98	110	124	138	175	208	243	278	313	350	416
6	25	32	38	44	51	57	64	70	76	88	102	114	128	160	191	223	255	286	318	382
6 1/2	23	29	35	41	47	53	59	65	70	82	94	106	118	145	176	206	235	264	290	352
7	22	27	33	38	44	49	54	60	65	76	88	98	109	136	164	191	218	246	273	328
7 1/2	20.4	25	31	36	41	46	51	56	61	72	82	92	102	125	153	178	204	229	250	306
8	19.1	24	29	33	38	43	48	52	57	66.9	76	86	96	120	143	167	191	215	239	286
8 1/2	18	22	27	31	36	40	45	49	54	62	72	80	90	110	135	157	180	202	220	270
9	17	21.2	25	30	34	38	42	47	51	60.1	68	76	85	106	127	149	170	191	212	254
9 1/2	16.1	20.1	24	28	32	36	40	44	48	56	64	72	80	100.5	121	141	161	181	201	242
10	15.3	19.1	23	27	31	34	38	42	46	54	62	68	76	95.5	115	130	153	172	191	230
11	13.9	17.4	20.8	24	28	31	35	38	41	48	56	62	70	87	104	122	139	156	174	208
12	12.7	15.9	19.1	22	25	29	32	35	38	44	50	58	64	79	95	111	127	143	159	190

are products of powder metallurgy, formed under pressure into compacts of the desired shape, then sintered (heated) into a dense and coherent metal. The cast alloys, as the term implies, are melted and poured into the desired form, even though the tungsten and sometimes the chromium carbides may be added to the molten carbide in the form of powders.

Until a short time ago it was considered impossible to machine these cast alloys after pouring by any means except grinding. Then it was found that certain sintered carbides, of the type used for turning hard cast iron, could be used for forming them. This discovery simplified the method of manufacture and made possible many cutting tools of more intricate shapes and applications.

It should not be inferred, because the discussion of the Co·Cr·W alloys is placed at the end of this book, that they are any the less (or more) valuable or widely used in industry than the sintered carbides. Their place in the cutting tool business is roughly midway between the so-called high-speed steels and the sintered carbides. This is shown by a study of Table I, which indicates that, in general, the cast alloys take up the work where high-speed steel leaves off, the sintered carbides coming into the picture where the cast alloys taper off. This table should not be considered the final authority on cutting speeds, however, because many operations today are being carried out at much higher rates than those shown. The figures given in Table I are highly conservative and, for the most part, represent starting points only. Similarly, it should not be inferred from the table that the place of the cast alloy in the cutting field ends exactly where the sintered carbide begins. Many heavy, high-speed cutting operations once considered only in the realm of the sintered carbides, are now being done with certain special types of cast alloys.

Cast Alloys As Tools. Haynes' first experiments were with alloys of nickel and chromium. His first successful mixture was an alloy of almost pure chromium and nickel, made by heating their oxides in an aluminum bath. This was a highly lustrous alloy and proved to be both malleable and ductile. With the proper heat treatment, it could be formed into wire or sheet. It was also practically insoluble in the common reagents, even in boiling nitric acid.

His success with this material caused Haynes to begin a series of experiments with cobalt, chromium, and aluminum. From this he got a few particles of a material which had much the same lustrous character as his first alloy, but which was considerably harder. Although it was malleable at a bright orange heat, it could not be worked to any appreciable extent. Haynes thought this metal would find wide use for tools and knives, particularly surgical instruments, which should not be easily corrodible. However, he discontinued work on it until about 1905 when he began experimenting with the idea of using the material for ignition points in automobile engines. More of the alloy was made, and it was found that in proportions of 75% cobalt and 25% chromium,

the alloy could be forged and even rolled to some extent, although with much difficulty. Haynes took out a patent in 1907 on cobalt-chrominum alloys for tableware, surgical instruments, and certain other uses, among which were cutting tools for fairly soft materials.

About 1909 or 1910, Haynes began a new series of experiments, adding tungsten and molybdenum to his cobalt-chromium base alloy. He found he had a material so hard that he decided to try it for the machining of cast iron. A boring mill operator was asked to test it on cast-iron pistons. The machinist running the mill shut it down at two o'clock in the afternoon, saying that he had done his day's work and had turned out his normal day's run of pistons! The new material had cut so much faster that production had been increased nearly 50 per cent!

Haynes named the new material "Stellite" because it was lustrous and nearly noncorrodible—similar to the stars he thought. Stellite, with the first letter capitalized, is today the trademarked name of one of the products made by the Haynes Stellite division of the Union Carbide and Carbon Corporation, although, as was pointed out earlier, the machine industry generally is inclined to refer to all these materials as "stellites," regardless of the manufacturer.

Properties of the Cast Alloys. It can be said in general that the cast alloys have three special properties. These are: first, they are untarnishable in ordinary atmosphere and resist attack by most common chemicals; second, while they are not as hard at room temperature as carbon-steel tools or even many tools of the newer alloy steels, they do have considerable hardness which ranges from about 375 to 650 Brinell, depending upon grade, make, and type; and third, they will regain their hardness an indefinite number of times even though operated at heats up to and beyond the point where high-carbon steel permanently softens or decarburizes.

There are a variety of these alloys for many purposes, a few of them containing infusions of iron or nickel to make them more easily workable. Some of them have found wide acceptance as materials for "hard-facing" almost anything of metal.

The Work They Do. The story is told that during World War I, a plant under contract to produce 9 1/2" shells at a rate of one each per minute for the British Government developed a bottleneck in the finish turning operation. With the tools then in use, new equipment would have been required to open up the choke point which would have thrown some 20 following operations out of gear. Tools of the new alloy were tested with the result that cutting time was reduced by about 50 per cent and production increased by about 30 per cent. This of course cleared up the congestion at that point. Piecework operators so valued the new tools that they carried them about in their pockets. Some even asked if it would be possible to buy the metal so they would be certain of higher daily pay.

The work done by Stellite, Tantung, Speedaloy, and the other cast alloys, whatever their trade names, in the war just ended would take

volumes to describe. A few typical jobs are illustrated in the following:

In the end milling of navy bronze (a pump part) a cut 3/8'' deep and 3/8'' wide had to be taken. With steel tools it was possible to turn out 3,750 pieces between grinds at a speed of 140 f.p.m. With cast alloy tools, production went up to 26,250 pieces between grinds and the rate to 260 f.p.m.

In boring a part made of 18-8 stainless steel with welded joints for a machine manufacturer, all tools, including the ones made of sintered carbide, broke at the brazed joint. A solid cast alloy tool bit was placed in the 20-year old boring head and made the rough cut at 240 f.p.m. The finishing cut was made at 346 f.p.m.

In turning a scaly, nickel-chrome steel forging having a Brinell hardness of 321, taking a cut of 1/8'' to 3/4'' deep at a feed of 1/16'', the best performance from high-speed steel tools was 45 f.p.m. with 23 lineal inches of stock being cut between grinds. The cast alloy cut 85 lineal inches of stock between grinds at a rate of 80 f.p.m.

In the cutting off of 18-8 stainless steel bar stock with a diameter of 2 13/16'' at a rate of 110 f.p.m. and a feed of .0045'', the cast alloy tool finished 154 pieces without resharpening and still did not appear dull, while the steel tool had to be resharpened every three pieces.

A cast alloy milling cutter having two inserted blades replaced a 9 tooth, 3'' diameter, high-speed-steel slab mill in the cutting of a dural sand casting of 200 Brinell hardness. The speed was 1,800 f.p.m. and the depth of cut 1/8''. The speed was tripled, the feed quadrupled, and the time between grinds increased from one to two hours with steel to 12 hours with the cast alloy blades.

Experimentation Continues. Not all such records have been set with simple Co·Cr·W alloys, however. In recent months, some manufacturers of these products have added other ingredients, notably boron carbide, to their mixtures. One manufacturer has for years added a proportion of tantalum. As a matter of fact, the examples just cited of the advantages gained through the use of the cast alloys were accomplished with this product. Some, without doubt, are using both tantalum and titanium, just as are the manufacturers of the sintered carbides. There is one ferrous alloy for tools that uses cobalt for "red hardness" and boron carbide for wear.

Except for the fact that the cast products contain chromium, most of the cast and the sintered carbides are a good deal alike, chemically. The cast alloys will perform many of the jobs done by the sintered carbides, and, in an occasional case such as that concerning the boring job, will do them better. However, it cannot be said that the cast alloys will do all the things the sintered carbides will do, or do them as well, for, after all, the cast alloys can be cut by the sintered carbides. Without going into any further technical detail, that alone tells much of the story. The cast alloys have their very definite place in the cutting tool field, as shown in Table I, but they are no cure-all and the best of them do not anticipate or replace the sintered carbides. After all, the sin-

tered carbides are the fastest, most efficient tools for general shop work man has ever devised.

A Metallurgical Analysis. The peculiar property of "red hardness" possessed by the cobalt-chromium-tungsten system of alloys is still not fully understood by metallurgists. That the cobalt imparts some special quality to the metal beyond acting as a mere binder has been fully demonstrated in the laboratories of every manufacturer of the cast alloys as well as the sintered. Exhaustive tests have shown that all these cast alloys last longer under abrasion and heat than many of those with higher hardness numbers, and sometimes even longer than the sintered carbides. When hardness is measured at room temperature by the usual methods, whether Brinell, Rockwell, or scleroscope, the cast alloys do not measure up quite as high as the hardened high-speed or the hardened high-carbon steels. Their value, however, as pointed out before, lies in the fact that the cobalt alloys have no "critical points" as do the iron-carbon metals. That is, they do not transform at certain given temperatures. They cannot be tempered. Once cast, they cannot be softened until they have been heated almost to the melting point—about 2,300° F.

It is true, of course, that these alloys, like all metals, do soften somewhat when heated, but they do it so slowly that at heats above 1,100° F. the tool grades are harder than any steel. The hardness of high-carbon steel drops off very rapidly above 400° F., while the best high-speed steels drop off about 950°. The "stellites," on the other hand, hold their hardness until 1,100° F. has been exceeded. Most of them maintain a considerable degree of hardness even at a cherry red heat, which is about 1,500° F.

Best of all, when these alloys are allowed to cool without quenching, they regain their original hardness. Tests in comparison with high-speed steel have shown that after a heating of half an hour at 950° F., most high-speed steels recover their hardness almost completely. However, if the same steel is heated to 1,100° F. or more, it must be retempered. The cast alloys will recover their original hardness after being heated at any temperature up to 1,850° F.

Although cobalt and nickel belong to the same chemical group and are considered to be very similar, chemically, they are vastly different in their ability to confer this quality of red hardness. Exaggerated stories came out of Germany at the end of World War II to the effect that tools made of nickel and titanium-vanadium carbides had kept Germany producing, and that they were likely to bring about new and even more revolutionary methods when introduced to the American tool industry. This would appear, on the face of the official report, not to be true. The Germans used nickel for a binder because they didn't have or couldn't get cobalt. They used titanium and vanadium substitutes because they couldn't get enough tungsten. In combinations, tungsten and titanium carbides serve excellently. But titanium carbide alone or even in high percentages never has and probably never will do the work of tungsten

carbide, much less with nickel as a matrix. American laboratories as as far back as Haynes' in 1899 have experimented extensively with these systems. Nickel is often used today as part of the binder for tools and materials for general purposes, and the Germans were able to use it as a substitute when they could do no better. However, it is not the metal for tools that have to "take it" day in and day out like those with a tungsten-cobalt base.

Special Cobalt Properties. Cobalt not only confers red hardness, but also adds to the compound's great resistance to wear and to its high strength, even at elevated heats. These qualities are probably the result of the peculiar grain structure developed by these alloys which shows, under proper magnification and etching, that the matrix consists not of cobalt alone, but of cobalt and some other constituent, probably a highly complex eutectic carbide. Sources for further study in the metallurgy of cobalt are listed at the end of the chapter.

If nickel is used to replace some of the cobalt, it has a definite softening effect on these alloys. However, nickel, iron or perhaps some other metals are often found as impurities, while, as stated before, iron or nickel sometimes are added for special purposes.

There are a number of metallurgical oddities about these alloys, most of which will not be discussed here for they are of interest primarily to the technician only. Anyone accustomed to dealing with tungsten can furnish a long list of its freakish characteristics. One such oddity, however, is worth examining.

Hardest Carbide Is Cast. The hardest tungsten carbide made is produced by the casting process. When the tungsten content is increased to more than 80 per cent and the carbon to more than 4 per cent, and when there is no chromium present in the mixture, a tungsten carbide results that is known as "diamond substitute." This material is able to take the place of industrial diamonds in many fields, particularly in oil well drilling and in certain types of hard rock mining. Small bits of cast tungsten carbide are welded into the cutting edges of rotary drilling and coring tools and on the surfaces of such devices as shovel dipper teeth. Crushed tungsten carbide in a steel binder is then welded over the surface to give support to the inserts. This is a different process than "hard facing" in which various cast alloys are welded directly to the surface or edge to be protected.

In the harder grades, principally those used for cutting tools, the Co·Cr·W alloys have practically no ductility at all. Like the sintered carbides, they break at about the same point at which they begin to "stretch." They do, however, take an extremely high polish which is useful in the burnishing of other metals and in the formation of parts requiring a high degree of finish.

Summary. To sum up, then, the qualities of the cast alloys can be simply stated thus: they are not as tough as high-speed steel, and they are not as brittle as the sintered carbides. They are particularly valuable in the cutting of materials that generate high heat in machining such

TABLE II. STOCK SIZES IN INCHES OF SOLID, SQUARE TOOL BITS

Width and Thickness	Length	Width and Thickness	Length	Width and Thickness	Length
3/16	3	7/16	3	5/8	4 1/2
1/4	2 1/8	7/16	3 1/2	5/8	5
1/4	3	7/16	4	5/8	6
5/16	2 1/4	1/2	2 1/2	3/4	3
5/16	2 1/2	1/2	3	3/4	3 1/2
5/16	3	1/2	3 1/2	3/4	4
5/16	3 1/2	1/2	4	3/4	4 1/2
3/8	2	1/2	4 1/2	3/4	5
3/8	2 1/2	1/2	5	3/4	5 1/2
3/8	2 3/4	1/2	6	3/4	6
3/8	3	5/8	3	7/8	5
3/8	3 1/2	5/8	3 1/2	7/8	6
3/8	4	5/8	4	1	5
3/8	6			1	6

as manganese, stainless, and other tough alloy steels. They are also valuable in the turning, facing, milling, and boring of cast and malleable iron, hard rubber, bronze, fiber, the plastics, and the light metals, at higher speeds than are possible with high-speed-steel tools.

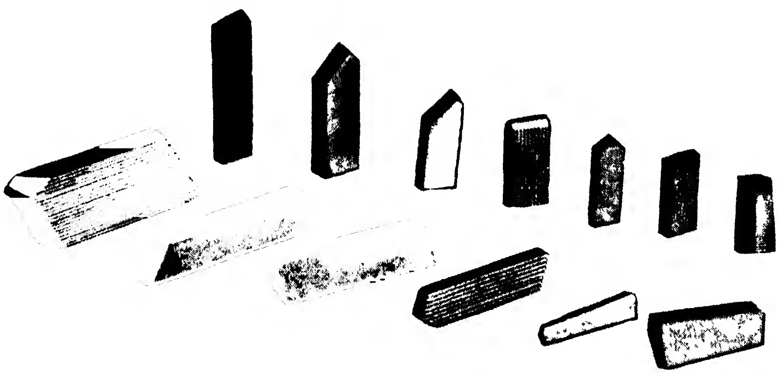


Fig. 1. A Sampling of the Tool Bits Available in the Cast Alloys
Courtesy of the Haynes Stellite Company

Cast Alloy Tools. In general, the cast alloy tools come in much the same standard forms, sizes, and shapes as do the sintered carbide tools. There is one outstanding difference, however. Turning tools up to 1" square are usually supplied and purchased in the solid carbide

TABLE III. STOCK SIZES IN INCHES OF SOLID, RECTANGULAR TOOL BITS

Width	Thickness	Length	Width	Thickness	Length
3/16	3/4	4	3/8	1	3
1/4	5/16	1 1/2	3/8	1	4
1/4	3/8	2	3/8	1	4 1/2
1/4	3/8	6	1/2	5/8	3
1/4	1/2	4	1/2	5/8	4
1/4	1/2	6	1/2	5/8	5
1/4	3/4	6	1/2	3/4	3
5/16	1/2	3	1/2	3/4	4
5/16	1/2	6	1/2	3/4	5
5/16	5/8	4	1/2	3/4	6
5/16	3/4	6	1/2	1	3
5/16	1	4	1/2	1	4
5/16	1	6	1/2	1	5
3/8	1/2	2	1/2	1	6
3/8	1/2	3	5/8	3/4	3
3/8	1/2	4	5/8	3/4	4 1/2
3/8	1/2	6	5/8	3/4	5
3/8	5/8	3	5/8	3/4	6
3/8	5/8	4	5/8	1	4 1/2
3/8	3/4	3	5/8	1	6
3/8	3/4	4	5/8	1 1/4	4
3/8	3/4	6	5/8	1 1/4	6
3/8	7/8	5	3/4	1	5
3/8	1	2 3/8	3/4	1	6

form rather than as a steel shank with a brazed-on carbide tip. Solid square bits are available generally in the sizes shown in Table II.

Rectangular tools up to 3/4'' × 1'' in cross section likewise are available and are often used as solid bits. These are generally available in the sizes shown in Table III.

Round tool bits, furnished centerless ground for use in turning and boring or for forming into special types and shapes, are usually available in sizes up to 1'' diameter. Common stock sizes are shown in Table IV.

The variety of shapes and sizes is perhaps even greater among the cast alloys than the sintered metals. A wide variety of boring and reaming blades, for example, is regularly supplied by the makers of these tools. There are also three or more styles of grooving tools regularly furnished by most manufacturers in this field, and a dozen or so styles of insert blades for milling cutters are commonly stocked, made for use in various makes of "standard" machines. A small sampling of these tips can be seen in Fig. 1.

TABLE IV. STOCK SIZES IN INCHES OF SOLID, ROUND TOOL BITS

Diameter	Length	Diameter	Length	Diameter	Length
1/8	2	7/16	4	11/16	3
3/16	2	7/16	5	11/16	4
3/16	3	7/16	6	11/16	5
3/16	4	1/2	3	11/16	6
1/4	2	1/2	4	3/4	3
1/4	3	1/2	5	3/4	4
1/4	4	1/2	6	3/4	5
5/16	2	9/16	2	3/4	6
5/16	3	9/16	3	7/8	3
5/16	4	9/16	4	7/8	4
3/8	2	9/16	5	7/8	5
3/8	3	9/16	6	7/8	6
3/8	4	5/8	3	1	3
3/8	5	5/8	4	1	4
7/16	2	5/8	5	1	5
7/16	3	5/8	6	1	6

In addition to these, the variety of tips is considerably greater than is commonly true of the sintered carbides. The same is true of the "standard" tipped tools. Fig. 2 shows several types of these tips. The design of 13 styles of tips which are commonly stocked is shown in Tables V through XVII inclusive, along with data giving the dimensions of

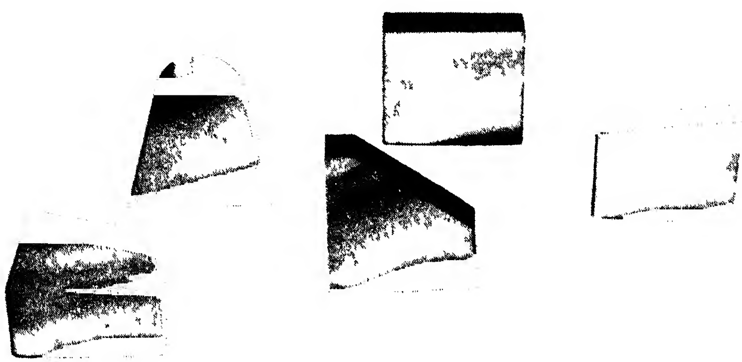
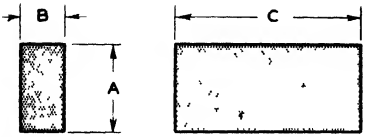


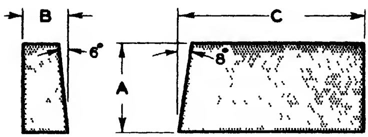
Fig. 2 A Few of the Many Standard Tool Tips Available in the Cast Alloys
Courtesy of the Haynes Stellite Company

TABLE V. DIMENSIONS FOR STYLE "A" TIPS



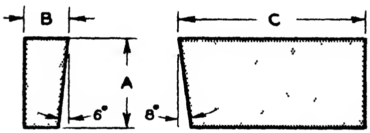
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	5/16	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	1/2	1 3/4
1	5/16	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE VI. DIMENSIONS FOR STYLE "D" TIPS



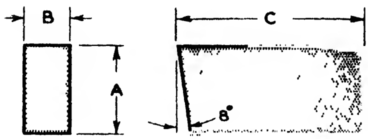
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	11/32	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	9/16	1 3/4
1	11/32	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE VII. DIMENSIONS FOR STYLE "G" TIPS



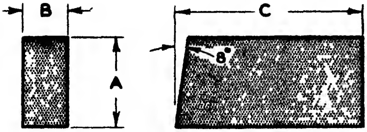
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	11/32	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	9/16	1 3/4
1	11/32	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE VIII. DIMENSIONS FOR STYLE "J" TIPS



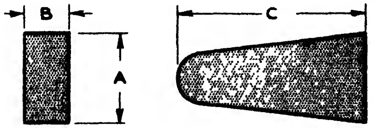
Width A	Thickness B	Length C
1/2	5/16	3/4
5/8	5/16	13/16
5/8	3/8	13/16
3/4	1/2	1 1/8
7/8	1/2	1 3/16
1	5/16	1
1	3/8	7/8
1	1/2	1
1	5/8	1 3/8
1 1/4	3/8	1 1/8

TABLE IX. DIMENSIONS FOR STYLE "K" TIPS



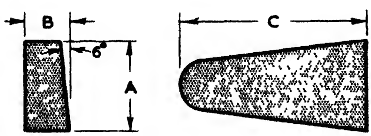
Width A	Thickness B	Length C
1/2	5/16	3/4
5/8	5/16	13/16
5/8	3/8	13/16
3/4	1/2	1 1/8
7/8	1/2	1 3/16
1	5/16	1
1	3/8	7/8
1	1/2	1
1	5/8	1 3/8
1 1/4	3/8	1 1/8

TABLE X. DIMENSIONS FOR STYLE "L" TIPS



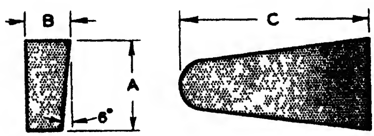
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	5/16	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	1/2	1 3/4
1	5/16	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/2

TABLE XI. DIMENSIONS FOR STYLE "M" TIPS



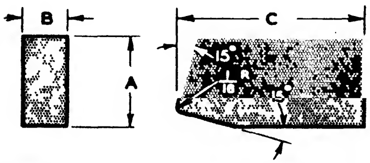
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	11/32	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	9/16	1 3/4
1	11/32	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/2

TABLE XII. DIMENSIONS FOR STYLE "N" TIPS



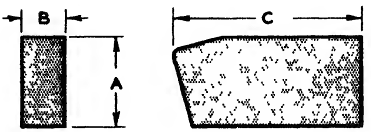
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	11/32	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	9/16	1 3/4
1	11/32	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/2

TABLE XIII. DIMENSIONS FOR STYLE "P" TIPS



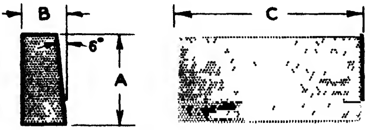
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	5/16	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	1/2	1 3/4
1	5/16	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE XIV. DIMENSIONS FOR STYLE "Q" TIPS



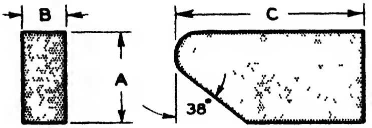
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	5/16	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	1/2	1 3/4
1	5/16	1 1/4
1	3/8	1 1/4
1	1/2	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE XV. DIMENSIONS FOR STYLE "V" TIPS



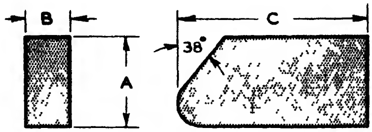
Width A	Thickness B	Length C
1/2	5/16	1 1/4
5/8	11/32	1 1/4
5/8	3/8	1 1/2
3/4	1/2	1 3/4
7/8	9/16	1 3/4
1	11/32	1 1/4
1	3/8	1 1/4
1	11/32	1 3/4
1	5/8	2
1 1/4	3/8	1 1/4

TABLE XVI. DIMENSIONS FOR STYLE "X" TIPS



Width A	Thickness B	Length C
1/2	5/16	7/8
5/8	5/16	1
5/8	3/8	1 1/8
3/4	1/2	1 3/8
7/8	1/2	1 5/8
1	5/16	1 1/8
1	3/8	1 1/4
1	1/2	1 1/2
1	5/8	1 7/8
1 1/4	3/8	1 3/8

TABLE XVII. DIMENSIONS FOR STYLE "Y" TIPS



Width A	Thickness B	Length C
1/2	5/16	7/8
5/8	5/16	1
5/8	3/8	1 1/8
3/4	1/2	1 3/8
7/8	1/2	1 5/8
1	5/16	1 1/8
1	3/8	1 1/4
1	1/2	1 1/2
1	5/8	1 7/8
1 1/4	3/8	1 3/8

the various sizes of each tip. Tipped tools are also available in the cast alloys line and include the styles usually supplied by manufacturers of sintered carbide bits as well as round nosed tools, tools offset or bent to angles other than 90°, and tools with double offsets. A number of standard tools which are readily available in sintered carbide or cast alloy is shown in Fig. 3.

In general, the shanks supplied for cast alloy tools are longer than those supplied with corresponding styles of sintered carbide-tipped bits. Careful comparison, style for style, will also show that longer, wider, and thicker shanks are also common with cast alloy tipped tools. To make this point clear, a common line of sintered carbide tools should be compared with a well-known line of the cast alloys.

A "style A" tipped tool, which is the simplest form, ground top, end, and side to provide the various reliefs, made by Carboloy (sintered carbide) is available in 17 sizes, beginning with 1/4" x 1/4" x 1 1/2" and ranging up to 1 1/2" x 2" x 8". The same type of tool in the Stellite (cast carbide) line is available in 10 sizes, ranging from 1/2" x 1" x 7" up to 1 1/4" x 1 1/4" x 7". The longest standard shank in the sintered carbide line is 8". The longest in the cast carbides is 12". It should be noted particularly that the smallest width among the cast carbides is 1/2", while the sintered carbide types begin with 1/4" widths. Likewise, the starting height with the cast carbides is 1", while 1/4" is the smallest height available in the sintered carbide tools. This, of course, is because the smaller sizes among the cast alloys are furnished in solid

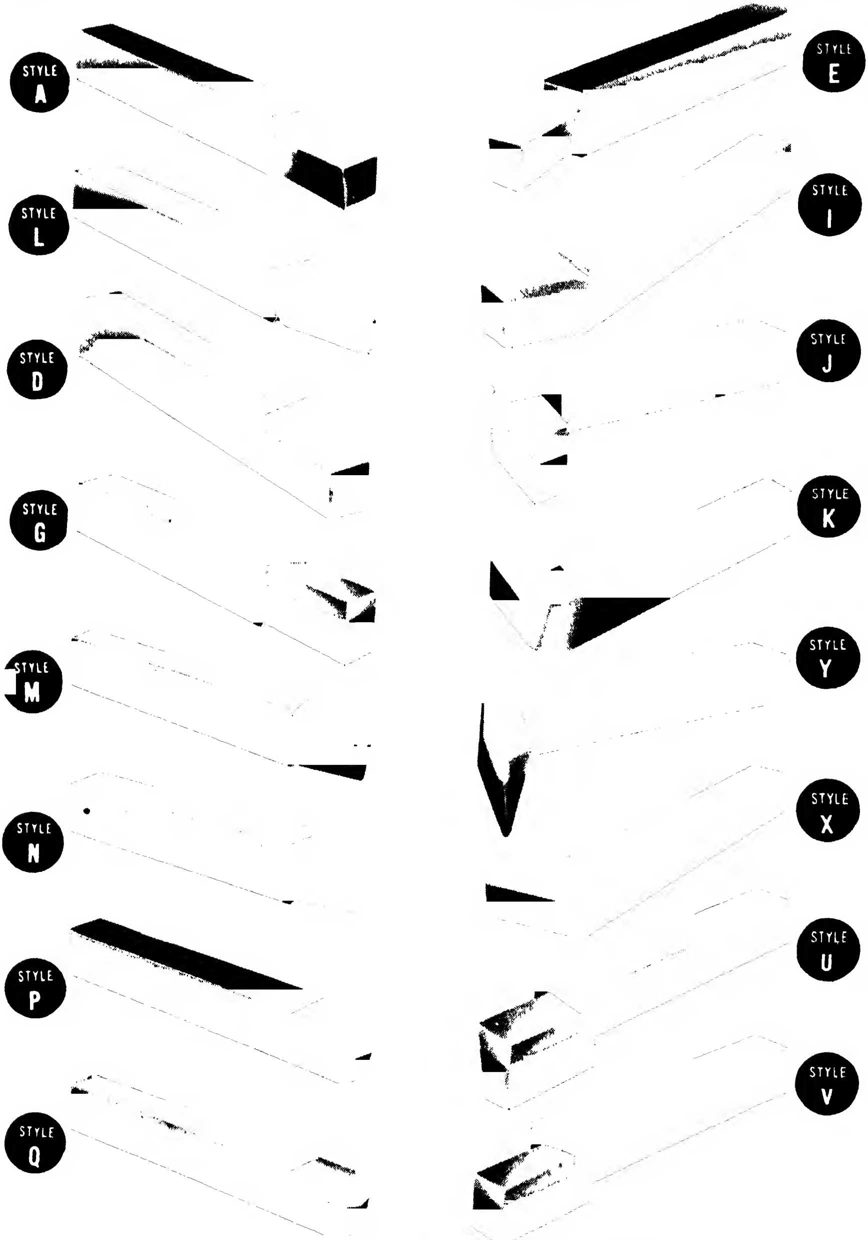


Fig. 3. Standard Cast Alloy Tools Which Are Factory Assembled, and a Variety of Some of the Available Tips

Courtesy of the Haynes Stellite Company

TABLE XVIII. COMPARISON OF STOCK, TIPPED TOOL SIZES

Shank Dimensions

Sintered Carbide			Cast Alloy		
Width	Thickness	Length	Width	Thickness	Length
1/4	1/4	1 1/2			
5/16	5/16	2 1/4			
3/8	3/8	2 1/2			
1/2	1/2	3 1/2	1/2	1	7
5/8	5/8	4	5/8	1	7
			5/8	1 1/4	8
3/4	3/4	4 1/2	3/4	1 1/2	9
			7/8	1 3/4	9
1	1	7	1	1	7
			1	1 1/4	7
			1	1 1/2	10
			1	2	12
1 1/4	1 1/4	8	1 1/4	1 1/4	7
1 1/2	1 1/2	8			
1 1/2	1	7			
5/8	1	6			
5/8	1 1/4	8			
3/4	1	6			
3/4	1 1/2	8			
1	1 1/4	8			
1	1 1/2	8			
1 1/2	2	8			

form. The complete table of sizes for this style in both lines is given for the purposes of comparison in Table XVIII.

Complete tables of all standard types and sizes, with diagrams of shapes, will not be given here. This information is readily available from individual manufacturers who issue catalogs, usually twice each year. All that is intended to be shown here are the general trends. Furthermore, most standard shapes were described in detail in Chapter III. The only additions in the cast alloy lines are those offset or bent to angles other than 90° and those doubly offset, which require no additional explanation.

The cast alloys are supplied by most manufacturers in two grades, although some producers do offer more than that number. The first, or common grade, is the one that has been used for years for machining cast iron, malleable iron, the softer steels, bronze brass, aluminum, and plastics. The second, or special grade, is intended to take heavier cuts in these same materials. It has increased hardness and usually is

specially developed for the faster machining of all materials. One manufacturer makes a special, wear-resistant grade for the machining of aluminum, magnesium, and their alloys.

In addition to the single-point or single-blade tools described in the foregoing material, most companies supply various types of milling cutters, either complete with the alloy brazed to solid teeth, or cutter bodies to take inserted blades with alloy tips. All these have standard specifications as to size of hole, counterbore, and drive, so as to fit common arbor equipment. A shell end mill with cast alloy edges brazed in is shown in Fig. 4.

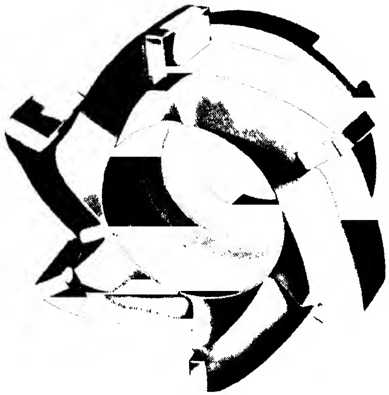


Fig. 4. Shell-Type End Mill Tipped with Tantung Cast Alloy
Courtesy of the Vascoloy-Ramet Corporation

Approximately the same variety of special tools is available in the cast alloys as in the sintered metals. Flat and circular forming tools, countersinks, spot facers, reamers, and specially ground bits for profiling work may be had on order. Some of these are illustrated in Fig. 5.

Making the Tools. In general, the methods used in tipping tools with the cast alloys are the same as those already given for the sintered carbides. One manufacturer does, however, recommend that a bronze brazing alloy be used to fasten the tips to the shanks or tools. It should be noted, though, that the manufacturer in question also makes large quantities of bronze alloy rod and flux for this and other purposes. Silver solder is most commonly used, copper being employed only where sandwich brazes are required.

If a bronze rod is used, it is recommended that after the shank recess is properly formed and the shank and tip have been cleaned as previously outlined, the tip as well as the shank should be tinned. After the tinning, the tip and the shank are placed together and heated either with the torch, in an atmosphere furnace, or brazed by any of the various induction methods. The torch method of brazing is presented in a step-by-step sequence in Fig. 6. This method will be found most useful when the user of these tools has small repairs to make or when the quantity of tools to be made is small, or when, in the case of milling cutters, the teeth are far enough apart that the brazing of one tooth will not melt out the braze on the previously brazed tooth. Furnace or induction methods of brazing will be found better under the same conditions as those given earlier in the case of the sintered carbides. Only one word of caution need be given in connection with the tipping of the tools. Always allow them to cool in the air. They should never be quenched.

Fig. 5. A Few of the Special Tools Available in the Cast Alloys
Courtesy of the Haynes Stellite Company

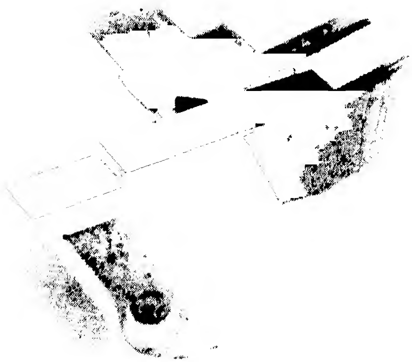
QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points in the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

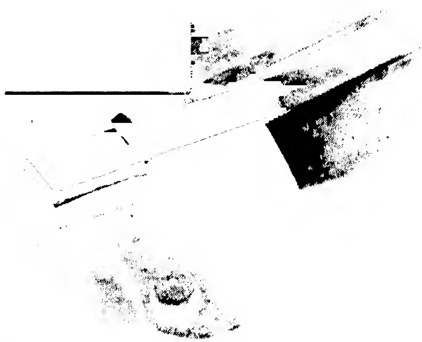
1. When was the first cast carbide made for cutting tools, and who made it?
2. What is a "cast carbide"?

Hone or file recess to remove burrs. Clean both blank and shank thoroughly (left).

The blank and shank ready for brazing (right).



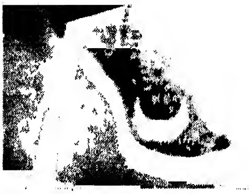
Place the shank in the vise, giving it a slight upward tilt (left).



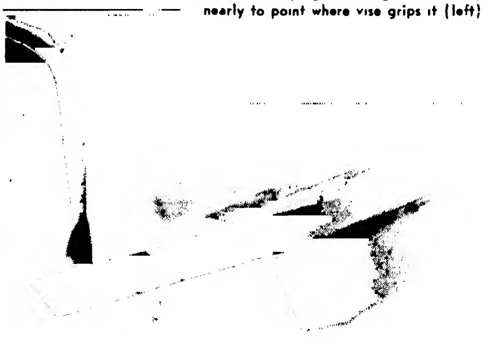
Lay brazing material and blank in place in the recess (right)

Fig. 6. On This and the Facing Page Is Illustrated the Method of Brazing Cast Alloy Tips to Tool Shanks

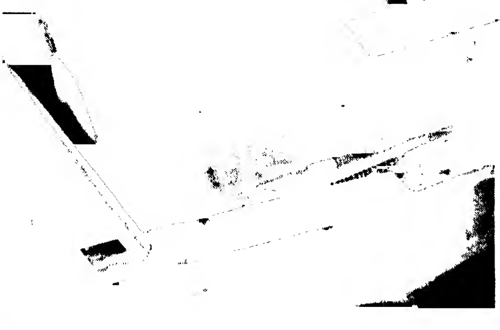
Adjust torch for a nonoxidizing flame (right).



Apply flame to bottom, sides, and end of shank keeping it moving. Heat evenly nearly to point where vise grips it (left)



After brazing alloy runs freely, apply flame to tip for a moment (right)



With poker or file move tip in proper position in recess (left)

With flame removed, hold tip in place with poker until braze has set. The braze should be allowed to cool slowly, the flux removed and the tool ground (right)

Fig. 6—*(Continued)*. This Is a Process Similar to That Described for the Sintered Carbides

3. How do they differ from the sintered carbides?
4. What does the cobalt do?
5. Can cast carbides be machined?
6. What is their place in the metal cutting field?
7. What are the special properties of the cast carbides?
8. Are they any more or less brittle than the sintered carbides?
9. Are they tougher or less tough than high-speed steel?
10. On what kind of work are they particularly valuable?
11. How do the cast and the sintered carbides differ chemically?
12. Can you temper a cobalt alloy?
13. Would you specify a cast carbide turning tool 1/4'' square in the solid or tipped form?
14. How is a cast carbide tool provided with a tip?
15. How should it be cooled when the operation is complete?

REFERENCES

Carboloy Co., Inc., The. Carboloy Tool Catalog, (GT-175-R). Detroit: The Carboloy Co., Inc., August 15, 1944.

Comstock, Gregory. "A Brief Summary of Certain Aspects of the German Hard Cemented Carbide Industry." Office of Publication Board, U.S. Department of Commerce, August, 1945.

Haynes Stellite Co. Haynes Stellite Metal Cutting Tools. Kokomo, Indiana: Haynes Stellite Co., February 1, 1945.

Vascoloy-Ramet Corp. Tantung Tool Catalog. North Chicago, Ill.: Vascoloy-Ramet Corp., 1943.

Wissler, William A. "Alloys of Cobalt, Chromium, and Tungsten," Metal Progress, August, 1939. Pp. 131-137, 163, 180.

CHAPTER XVII

Grinding and Use of Tools

Factors in Grinding the Carbides. The cast alloys having cobalt as a base may be hand or machine ground. The grinding may be done either wet or dry, just as was true of the sintered carbides. The determining factors in grinding depend on the individual and the equipment available. One general recommendation can be made, however, and that is that the wheel speed should range between a minimum of 2,800 f.p.m. and a maximum of 4,200 f.p.m. Any speed lower than 2,800 f.p.m. may result in "loading" of the wheel and the consequent need for redressing. Higher speeds and unusually heavy pressures are likely to cause checking of the carbide tip. As a means of facilitating finding the proper speed in r.p.m. of any diameter grinding wheel, based on its speed in f.p.m., Table I is presented.

The speed in surface feet per minute, or f.p.m., may always be found, however, by multiplying the r.p.m., the cutter diameter, and .2618.

For hand grinding, the tool should be applied to the wheel with moderate pressure and moved constantly across the face of the wheel. If the tool consists of a steel shank which has been tipped with cast alloy, the shank may be cooled in water at intervals, if this seems essential. Normally, such procedure should not be necessary because if the shank is too hot to handle, it is pretty good evidence that too much pressure is being put on the work. The entire tool must never be quenched, however, because such action is almost certain to cause surface checks which will eventually lead to tool failure. While many people do quench cast alloy tools in water during grinding with no apparent ill effects resulting to the tool, it is, nevertheless, poor practice and should be avoided. If the cutting edges are lapped or hand stoned after the finish grind, where both roughing and finishing grinds are used, the tool will be found to work better and longer.

Most manufacturers in this field recommend that their material be ground by machine, probably because they have found by bitter experience that few operators can be trusted to grind the tools by hand. When grinding the tools by machine, light cuts with a fast table traverse should be taken as was explained in Chapter V. The wheel feed will vary according to the size and shape of the tool, just as was true of the sintered carbides. These are matters which are pretty much up to the

TABLE I. R.P.M. OF GRINDING WHEELS OF VARIED DIAMETER, BASED ON THEIR SPEED IN F.P.M.

Wheel Diameter in Inches	R.P.M. at Established F.P.M. Speeds				
	2,700 F.P.M.	3,000 F.P.M.	3,500 F.P.M.	4,000 F.P.M.	4,500 F.P.M.
1..........	10,319	11,465	13,376	15,279	17,189
2..........	5,160	5,732	6,688	7,639	8,594
3..........	3,440	3,822	4,460	5,093	5,729
4..........	2,580	2,866	3,344	3,820	4,297
5..........	2,064	2,293	2,675	3,056	3,438
6..........	1,720	1,911	2,230	2,546	2,865
7..........	1,473	1,637	1,910	2,183	2,455
8..........	1,290	1,433	1,672	1,910	2,148
10..........	1,032	1,147	1,337	1,528	1,719
12..........	860	955	1,115	1,273	1,432
14..........	737	819	955	1,091	1,228
16..........	645	717	836	955	1,074
18..........	573	637	743	849	955
20..........	616	573	668	764	859
22..........	469	521	608	694	781
24..........	430	478	557	637	716

experience of the operator and the features of the machine being used. In general, however, the narrower the surface being ground, the heavier the feed may be.

If the grinding is done wet, an adequate amount of coolant should be used and should be directed at the point of contact at rather a low velocity.

Table II gives the recommended grades of grinding wheels for use with the cast alloys. The standard markings are those of the Grinding Wheel Manufacturers' Association. These recommendations are general and, if one has a special problem, it is always best to consult the manufacturer of the particular wheel being used. Often the same grade of wheel can be used for either of the two usual grades of cast alloy, but, in some cases, it will be noted that a different wheel is specified for Grade 2. Here, Grade 2 means the special, or steel, grades of cast alloy as distinguished from the "regular" or "standard" grade supplied by every manufacturer of these materials.

It is just as important with the cast alloys as with the sintered carbides that the tools be sharpened properly so that they have highly finished cutting edges, proper clearance angles, and no checks in the surfaces as the result of overheating. The wheels used, as will be noted from the table, are from one to three grades softer than those used to

TABLE II. GRINDING WHEELS FOR THE CAST CARBIDES

Type	Grade 1	Grade 2
Hand grinders		
Dry	A46-M5-V	A46-L5-V
Wet	A46-N5-V	A46-M5-V
Surface grinders		
Reciprocating table		
Coarse	White A46-15-V	White A46-H5-V
Fine	White A80-H5-V	White A80-H5-V
Rotary table	A46-G5-S	A46-G5-S
Tool grinders		
Universal grinders	White A46-J5-V	White A46-J5-V
Sellers type	White A46-J5-V	White A46-J5-V
Gisholt type	White A46-J5-V	White A46-J5-V
Ingersoll type	White A60-J5-V	White A60-I5-V
Cylindrical grinders		
Large diameter	A50-L5-V	A50-L5-V
Small diameter	A60-L5-V	A60-L5-V
Chip breaker grinding	A46-G8-B	A46-G8-B

sharpen high-speed-steel tools. The abrasive is aluminum oxide and the bond recommended is the new vitrified type. It follows, then, that the wheels used to grind the cast alloys will also grind high-speed steel, though less economically. However, the grades of the grinding wheels used for high-speed steel should not be used on the cast alloys because, normally, they are too hard.

Grinding Single-Point Tools. In hand grinding the cast alloys, it will be found a time saver if the grinding machine is equipped with a work-rest table that is adjustable to any clearance angle that is desired. If the grinding machine used does not have such a table, a template often can be rigged with which it is possible to check the tool profile quickly and easily.

Ordinary dull tools may be sharpened either on the periphery of the straight wheel, as in Fig. 1, or on the side of the cup or cylinder wheel, as in Fig. 2. Note the work-rest table in Fig. 2. The straight-hard wheel tends to undercut or "hollow grind" the clearance, or even the face of the tooth or bit, in the same manner as was noted in Chapters V and XI. This difficulty can be obviated by using a large diameter wheel or by holding the tool at a slight angle. Normally, however, the straight wheel will be used for the rough grind and the cup or cylinder wheel for the finish operation. Clearance angles are much more easily controlled with the cup wheel.

Fig. 1. The Ordinary Dull Tool May Be Ground on the Periphery of the Straight Wheel

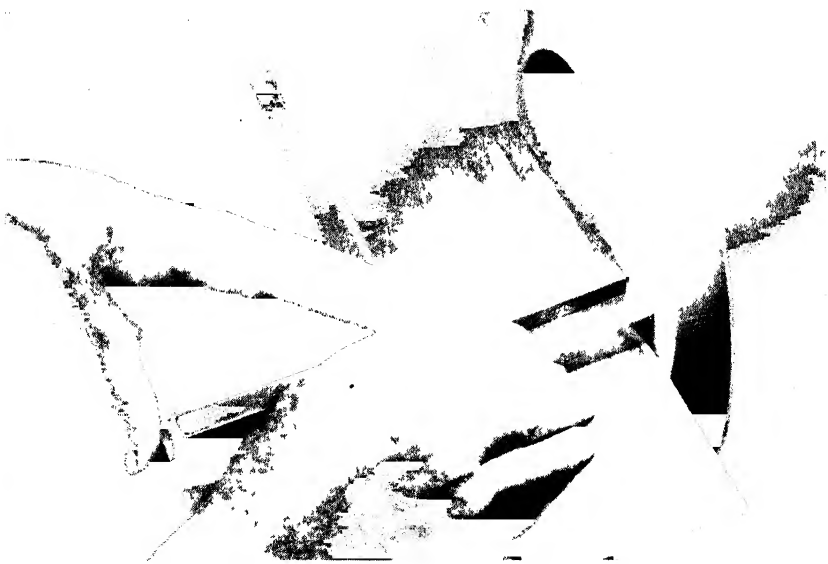


Fig. 2. Finish Grinding the Tool on the Side of the Cup Wheel
(Note the Use of the Work-Rest Table)

Grinding of the top face should be avoided as much as is possible, because this material is cast, and its best metallurgical structure lies nearest the original cast surface. In use on steel, of course, the cratering action of the tougher chips will make some touching up of the top surface unavoidable, but if the tool is given a high finish on its top face to begin with, much of this difficulty can be overcome. Chip breakers

Fig. 3. Keep the Tool on the Move across the Face of the Wheel. Here the Side Cutting-Edge Angle Is Being Ground Offhand on a Special Wheel
Courtesy of Science and Mechanics Magazine

can be and often are ground into the cast alloys just as was described for the sintered carbides. This point will be discussed in greater detail further in the chapter.

A revolving lap wheel is probably best for the finest finish of the top surface and the tool edge, although the finishing operation can be done with a hand stone, preferably an India oilstone.

Where the tools are badly chipped and broken, especially in the case of the smaller cast alloy tools which are made solid, the tools should first be ground off rapidly to get a fresh, straight surface from which to work. Moderate pressure and relatively light feed, however, must still be used. One way to achieve speed is to notch in the tool edge either with a thin cut-off wheel or with the edge of the straight wheel, then grind off the remaining surplus and proceed with the sharpening in the usual way.

If the tool is the tipped type, whether brazed with bronze, silver

solder, eutectic alloy, or copper, the grinding should be done alternately on the tip and the shank. Front and side clearances should be finish ground with the tool resting firmly on the table as shown in Fig. 2. Light pressures should be used and the tool should be kept constantly on the move across the face of the wheel. This procedure is shown in Fig. 3.

Everything that has been said heretofore concerning the need for frequent resharpening of the sintered carbide tool applies just as force-

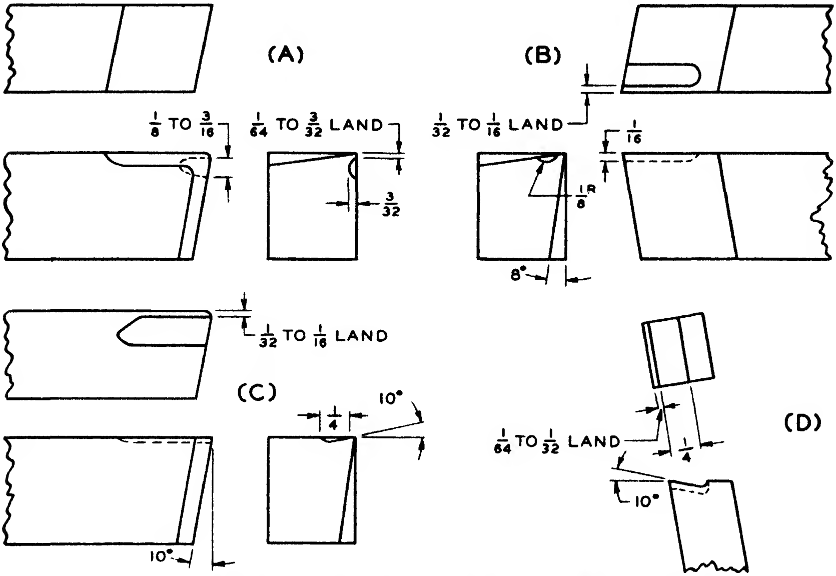


Fig. 4. Experience and Experimentation Have Shown These Chip Breakers To Be Best Adapted to the Cast Carbides

fully to the cast alloys. Likewise, the sharpening should be carefully done, preferably by an experienced operator. While the cast alloys normally are not so expensive as the sintered variety, their cost is greater than that of high-speed-steel tools. They must be handled properly in order to get the efficiency from them that is built in them.

Chip Breakers. There are four types of chip breakers that have been found, through much use and experimentation, to be best adapted to the cast carbides. These are shown at (A), (B), (C), and (D) in Fig. 4. Of these, the grooved lip style shown at (B) is the most commonly used. If the work is heavy, the land should be at least 1/16" in width.

Tools which are ground following the pattern shown in (C) are usually best for the turning of steel because they have a greater lip strength than when ground with a radius groove. The tool shown at (D) is an end-cutting turning tool which is grooved all the way across the top. The land can be narrower in this case because the cutting edge is better

supported than it is in the side-cutting tools. As was seen earlier, the width and the angle of the groove control the length and curl of the chip.

Although some of the cast alloys in use on steel machining operations have angles of 18°, and some as high as 22°, the 10° angle of the groove shown in (C) and (D) is generally satisfactory. In specifying the angle, the tool engineer must always take into consideration the kind of part being machined, the metal of which it is made, the depth of cut, the feed, and the finish.

The simple chip breaker shown in (A) of Fig. 4 is easily and quickly ground by the offhand method on the edge or the corner of the wheel. The grooved types, however, should be ground with precision on a surface grinder or on a tool and cutter grinder. The groove shown at (B) is ground with a 1/8", resinoid-bonded, cut-off wheel. For special forms and for wider grooves, vitrified wheels with faces rounded or shaped to conform to the form of the groove wanted should be used.

Inserted Blade Milling Cutters. As previously stated, the inserted blade milling cutter should be taken out of the machine and resharpened at the first sign of dulling. This not only reduces the rate of wear, but also makes necessary the removal of a lesser amount of material to put the cutter back in good condition.

There are two general types of grinding machines for inserted-blade milling cutters. On the Ingersoll and the Oliver automatic grinders and others of their type, the straight wheel is used. On the Milwaukee, Cincinnati, Norton, and similar machines, the cup-type wheel is used. The wheels recommended for use on these various machines are given in Table II.

The only general recommendation that need be given here in addition to those given earlier is this: before the cutter is rough ground, the grinding wheel should be diamond dressed to prevent it from glazing and heat checking the blades. It should also be redressed before the finish grind operation. This procedure was fully explained in Chapter V.

The clearance and relief angles to be given the cutter blades are usually a matter of design and depend upon the machine and the material to be cut, but in general it can be said that the cast alloys are normally ground with a primary relief of 2° to 6° on face, corners, and periphery. In many plants, however, it is the practice to grind a relief of 2 1/2° to 3° on the face and 5° to 6° on the corners and the periphery. The secondary clearance is usually 20° and 30° and, of course, should be ground first and as often as necessary to keep the land at 1/16" to 1/8". Light cuts should be taken, generally not more than .002" or .003" per pass. For roughing, a rapid traverse should be used, either of the table or the wheel, depending upon the machine. For finishing, the cut should be reduced so that not more than .0005" is removed. A slow traverse should be used, also. Some users take a first roughing cut of .003" followed by a semifinishing cut of about .0015". This is followed by the finishing cut of .0005". Typical setups for machine grinding are shown in (A) and (B) of Fig. 5.

The practice in grinding the cutter is the same as explained earlier

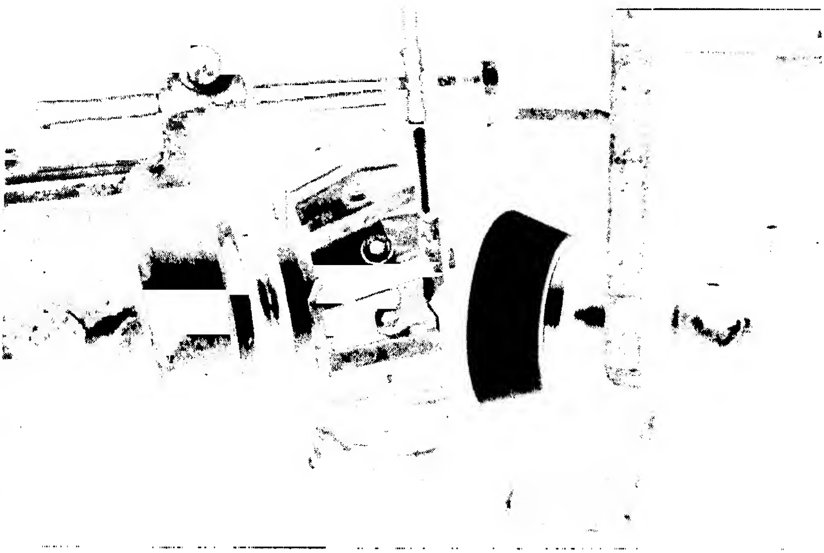


Fig. 5.(A) Setup of a Tool and Cutter Grinder for Grinding Corners on an End Mill
Courtesy of the Norton Company

Fig. 5.(B) Grinding the Peripheral Clearance with a Special Cutter Grinder
Courtesy of the Norton Company

in the sections on the sintered carbides. One can either grind completely around the cutter, then revolve it 180° and start the finish grind on a blade opposite the original starting point, as shown in Fig. 6, or the cutter can be divided into four equal parts or sections and marked accordingly with a piece of chalk. One section at a time is then rough ground, followed by the finish grind, until the job is complete. However, instead of progressing around the cutter in a continuous fashion, the section opposite the first is sharpened second in order. In other words, section 1 in Fig. 6 is sharpened first, followed by section 2. Section 3 is then sharpened, and the job completed by sharpening section 4. The mechanics who favor this method, sometimes called the "quadrant method," believe it gives a more accurate grind and a better finish to the work.

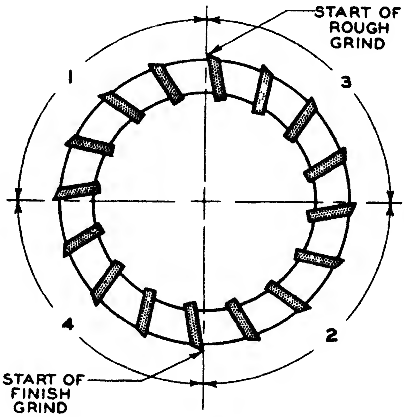


Fig. 6. The Quadrant Method of Grinding Milling Cutters

After the cutter has been sharpened, it should be inspected with a dial indicator setup as described in the section in this book on sintered carbides. The faces of the blades should be of the same height within .002" for roughing, and within .0005" for finishing work. A typical setup is shown in Fig. 7.

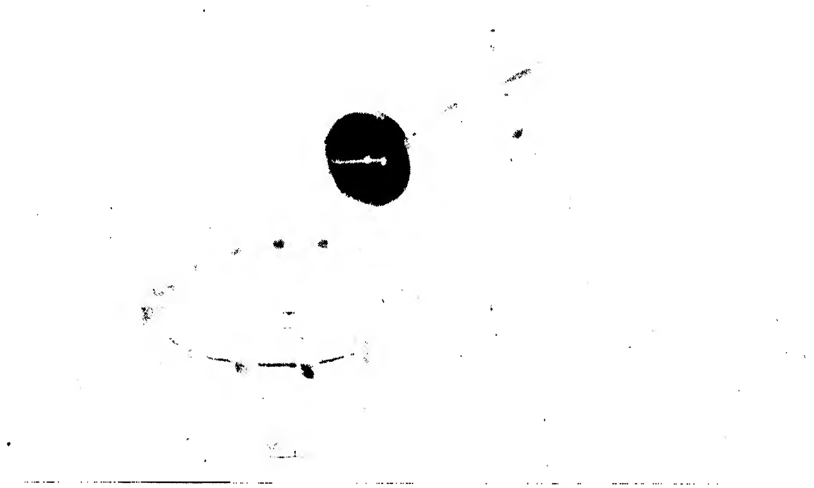


Fig. 7. Typical Setup for Grind Inspection, Using a Dial Indicator
Courtesy of the Norton Company

Sharpening Solid Cutters. Solid cutters, end mills, counterbores, reamers, and other tools of this type should be sharpened on the periphery, or the sides, by grinding the relief angle behind the cutting edge of each tool. Such a setup is illustrated in Fig. 8. Although a cup wheel is being used in this instance, a straight wheel might just as easily have been chosen.

The chief point to watch in using these tools is to keep the cutting edges radially equal, since a cutter or other tool of this type, when out

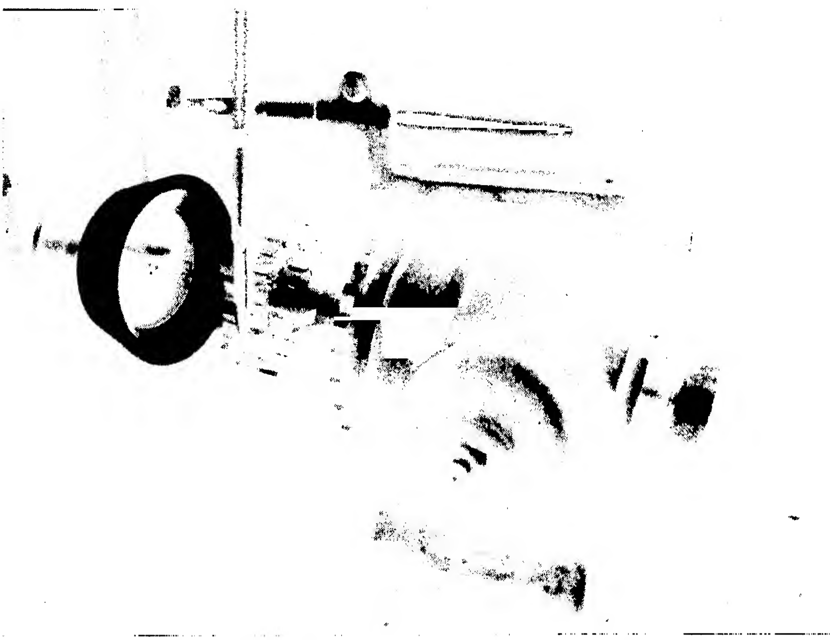


Fig. 8. Grinding the Face Clearance with a Tool and Cutter Grinder
Courtesy of the Norton Company

of true, runs with a constant pounding and hammering. This may ruin the cutter and perhaps the work. To make the edges equal, very light cuts should be taken which are not more than .002'' for roughing, nor more than .0005'' for finishing. The cutter should be revolved 180° after the first roughing and/or the first finishing cuts have been made, and the tooth directly opposite sharpened. The adjacent tooth is sharpened next, then the tooth 180° from it, etc., until all teeth have been sharpened.

After the teeth have been resharpened a number of times, it may be necessary to regrind the secondary clearance to reduce the width of the land to about 1/16''. This will prevent the heel of the tooth from dragging on the work.

Such tools as end mills, spot facers, reamers, and counterbores are all sharpened in much the same fashion as that just described for the milling cutters: by "backing off" the teeth to the proper clearance angle. Reamers, however, cannot be resharpened on the periphery, since in so doing they lose their size. When they become dull they can be ground to the next smaller size. The land left on reamers is usually 1/32", depending upon the size of the reamer and the work on which it is to be used.

Fig. 9. Setup for Sharpening a Dovetail Forming Tool, Using a Vise Fixture on a Small Surface Grinder

Courtesy of the Norton Company

Circular Forming Tools. Circular forming tools, just as in the case of the sintered carbide tools studied earlier, are sharpened by grinding on the cutting face so as to leave the original form unchanged. The method is exactly as explained in detail earlier in the book and requires no additional instruction. The downfeed of the wheel, however, should be held from .001" to .002" per pass for roughing these tools and from .00025" to .0005" for finishing. The cross feed and traverse both should be high.

Dovetail Forming Tools. The sharpening of dovetail forming

tool is, to all intents and purposes, a surface grinding operation. Only enough stock is removed from the end of the cutting edge to restore keenness. The method is the same as that described earlier, using any of the systems given in the section on sintered carbides. Clearance angles are usually 10° to 15°. The downfeed for the roughing operation should be no more than .002'' and for the finishing pass, no more than .0005''. Again, table traverse and cross feed should be high. A typical setup for sharpening dovetail forming tools is shown in Fig. 9.

Use of Tools. It should always be remembered that the job of any cutting tool is to remove metal as efficiently as possible. This may not always mean a high surface speed. As has been seen before, the place of the cast carbides is roughly between that of the high-speed-steel tools and the sintered carbides. This means that so far as surface speed is concerned, the cast alloys can be used at speeds which are 50 to 200 per cent higher than high-speed steel, and about half as fast as the sintered carbides.

Under ordinary conditions it will be found best for the operator to start with a feed and speed which is about double that usually used for high-speed-steel tools on the same operation. After the tool has been in service long enough to show some signs of wear, it should be examined with a magnifying glass. If there is too much wear on the front relief and but little on the top, the speed should be pushed up. If there is too much top wear, the speed should be lowered until the front and the top wear are about even. Sometimes this may be accomplished by a slight change in top rake or side rake without altering the speed or feed. However, this will usually depend on the material being cut.

A high surface speed need not necessarily reflect the efficiency of the tool. That is, the higher speed may not always guarantee the removal of the most metal. Actually, the linear travel of the tool, or the feed and the volume of material removed by the tool per sharpening, are the best criteria at the machine, of tool efficiency. Of course there are a great many other factors that enter into over-all tool efficiency which have been explained in preceding chapters. Such points as length of tool life, time between grinds, the finish obtained, and power consumed are all indications of tool performance.

It is necessary for the machine operator and the tool engineer to co-ordinate speeds, feeds, dimensional accuracy, and the depth of cut consistent with good finish obtained in order to achieve maximum production. Production is the machine operator's principal job, while the tool engineer is concerned with efficiency. If there is lacking the proper balance between feed, speed, and depth of cut, there certainly will be a loss of production which will reflect strongly on the efficiency of the tool and its design.

A number of special tool holders have been developed for the cast alloys. These tool holders have been found highly useful and responsible for great savings in cost. They keep the tool in proper relation to the work, helping to make certain the machine and the tools do the best job

possible under any given set of conditions. These factors are especially useful in the cast alloy field since most of the smaller sizes of cast alloy bits are produced in solid form, calling for especially short overhangs and rigid machine conditions.

Some of the toolholders, or adapters, which are typical of the many varieties available, are shown in Fig. 10. These holders are designed to give the proper support to the solid tool for boring, facing, turning, recessing, grooving, cutting off, and all the other operations for which such small, single-point tools are used. The design of the holder will suggest its application. Note that in the toolholders in (A) and (B), the

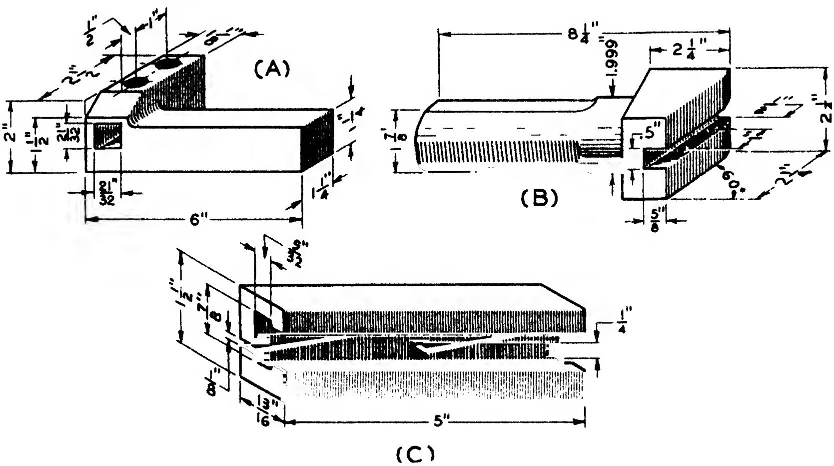


Fig. 10. A Representative Selection of the Tool Holders or Adapters Available for Use with the Cast Alloys

setscrews have been placed as near the end as possible so that the lengths that normally would become scrap can be used still longer. There are, of course, other designs that will achieve the same effect. These are merely shown as a suggestion.

In (C) of Fig. 10 is shown a toolholder which is designed for holding tools in parting and grooving operations, and which is particularly valuable where it is necessary to work with tools that must be used close against a shoulder.

Toolholders similar to those shown have been developed which are actually tool adapters, the purpose of which is to enable the use of solid bits in toolholders which originally used bits of a larger size. Other holders are particularly useful in adapting older machines to the use of the cast alloys. Similar holders have been devised which were successful in reducing tool bit cross sections where it was desired to use the smaller square and rectangular solid cast alloys on machines that originally had been intended for the larger shanked, high-speed steel or

TABLE III. RECOMMENDED RELIEFS FOR SINGLE-POINTED CAST ALLOY TOOLS

Angle	Steel	Cast Steel	Stainless Steel	Cast Iron	Bronze
Side rake	8-20*	8	8-20*	4	4
Side cutting-edge angle	10	10	10	10	10
Side relief	7	5	7	5	5
Back relief	8-20*	8	8-20*	0	4
End cutting-edge angle	15	10	15	10	10
Front relief	7	5	7	5	5

*Figures give the range. The angle depends on the type of steel. In general, the harder the steel, the smaller should be the rake angle.

sintered carbide bits. Many plants solve the problem of converting to a product much smaller or entirely different from anything contemplated when the machine was bought, by using a variety of these toolholders. Odd as it may seem, the changeover to smaller tools often results in more pieces per grind. These adapters may be made up either right- or left-handed and allow the use of 1/2'', 5/8'', or 3/4'' bits in holders intended for 3/4'', 7/8'', 1'', or even 1 1/4'' shanks.

Unless the conditions of the work and the material call for the use of tipped tools with larger shanks, it will often be found that the solid types are the most economical. Not only are many more grinds possible with the solid bit, but there is no fabrication cost as in the case of the tipped tool. Above all, however, a fine tool should never be condemned because a poor toolholder was chosen to be used with it. The toolholder itself may be so badly worn that it does not offer adequate support and causes the tool bit to crack.

Rigidity Essential. Just as in the case of the sintered carbide tools, deflection of the cast alloys must be held to the absolute minimum. The overhang, therefore, must be kept as short as possible, and the surface on which the tool rests should be absolutely flat. It cannot be rocked in a high-speed-steel type of toolholder any more than can the sintered carbide tool. There must be no ''spring'' in the work either. Also, just as with the sintered carbides, the tool must be kept at the proper position against the work so that all the cutting and clearance angles designed into the tool are maintained. If it is found necessary, the nose of the tool can be set a little above the center line of the work. The effective cutting angles used with the cast alloys are usually a little less than those recommended for the sintered carbides. For roughing cuts, a larger nose radius than that recommended for the sintered carbides is often better.

Recommended Speeds and Feeds. Cast alloy single-point tools normally will be ground with relief angles of no more than 7°. This is done to give maximum support to the cutting edge and applies whether the tools are tipped or are solid. Rake and cutting edge angles, of course, will vary with the material being machined. Table III will give any operator a start in the right direction. The angles shown in the table are for cutting tools. In general, boring tools will use the same angles, except that for small holes, clearance angles must be great enough to allow the boring bar to clear the work.

The speeds shown in Tables IV and V are given for various feeds and materials and, as before, represent nothing more than average figures. Any special conditions will entail experimentation or the good judgment of long experience to determine the special feeds and speeds required.

It may be found that these "starting points" will be but half the cut it is possible to take. If the depth of the cut is less than that shown, a proportional increase can usually be made in the surface speed or the feed, or sometimes both. If the cut is intermittent—a condition which would be unusual in turning or facing operations—the speed will have to be lower than that shown or the feed will have to be reduced. The footnotes give additional special information.

Boring operations are often carried out together with other machining. When this is the case, the factor governing the whole operation is usually the speed at which the other operations are done. If a part is to be turned and bored at the same time, for example, the surface speed chosen for the turning would also govern the speed for the boring job. However, unless boring operations enter the picture, speeds should be somewhere near the figures given in Tables IV and V.

Multiple-Pointed Tools. Since the most common application of the multiple-pointed tool is found in milling, recommendations for feed and speed of milling cutters fitted with cast alloy cutting edges will be given here.

The axial rake, often called merely, the rake, and the radial rake, often termed the hook, are the back and side rake, respectively, of the single-point tool, translated into rotary terms. This was described in detail in Chapter X. As in the case of the single-point tools just considered, these angles on the milling cutters which are equipped with cast carbide teeth, depend upon a great many conditions. Primarily, however, they depend on the material to be machined.

Normally, it is recommended that relief angles for milling cutters be kept at about 6° so as to provide the greatest amount of support. This rule, though, does not hold good for aluminum, magnesium, and the other light metals. Many shops use relief angles up to 10° on these materials. If more clearance than that is required, it should be ground from the heel of the blade. The blades should always be wedged or clamped into the cutter, but never driven or forced into place.

As a means of refreshing the memory, details of the axial and radial

TABLE IV. CUTTING SPEEDS IN F.P.M. FOR CAST ALLOYS FOR NONFERROUS METALS, CAST AND MALLEABLE IRON AND CAST STEEL

Material Cut	Feed in Inches	Cutting Speed for Depth of Cut				
		1/32 to 1/16	3/32 to 3/16	7/32 to 5/16	11/32 to 1/2	17/32 to 5/8
Aluminum and Magnesium	.006-.015	1100	1000	900	800	700
	.016-.031	1000	900	800	700	600
	.032-.062	900	800	700	600	500
Brass (yellow)	.006-.015	550	500	450	400	350
	.016-.031	500	450	400	350	300
	.032-.062	450	400	350	300	250
Bronze (hard)	.006-.015	130	115	100	90	80
	.016-.031	115	100	90	80	70
	.032-.062	100	90	80	70	60
Copper	.006-.015	450	375	350	275	240
	.016-.031	400	325	300	240	200
	.032-.062	350	275	250	225	175
Cast Iron (soft)	.006-.015	220	200	175	160	125
	.016-.031	200	180	150	140	115
	.032-.062	180	160	125	115	90
Cast Iron (medium hard)	.006-.015	175	150	125	100	90
	.016-.031	150	125	100	85	70
	.032-.062	125	100	90	70	50
Steel Castings (soft)	.006-.015	175	150	125	100	95
	.016-.031	150	125	115	90	80
	.032-.062	125	100	95	75	60
Steel Castings (medium hard)	.006-.015	160	140	120	100	85
	.016-.031	140	120	100	90	75
	.032-.062	120	100	85	80	50
Malleable Cast Iron (soft)	.006-.015	300	250	225	200	180
	.016-.031	250	225	200	180	160
	.032-.062	225	200	180	160	140

NOTE. The cutting speeds given in the table are designed for short cuts and the use of coolants during the cutting action.

For continuous cuts without the use of a coolant, the cutting speeds should be reduced 15 to 20 per cent.

For intermittent cuts where a coolant is used, the cutting speeds should be decreased 15 to 20 per cent. Where no coolant is used, the speed should be reduced 20 to 25 per cent.

For light finishing cuts, where feeds are from .002" to .004" per revolution, the speed may be increased 40 to 80 per cent.

TABLE V. CUTTING SPEEDS IN F.P.M. FOR CAST ALLOY TOOLS ON CARBON ALLOY AND STAINLESS STEELS

Material Cut	Feed in Inches	Cutting Speed for Depth of Cut				
		1/64 to 1/16	3/32 to 3/16	7/32 to 5/16	11/32 to 1/2	17/32 up
S.A.E. X1112	.004-.015	325	250	200	190	175
	.016-.031	250	225	180	175	150
	.032-.062	200	190	175	150	100
X1112	.004-.015	225	190	170	150	130
S.A.E. X1314	.016-.031	190	170	150	125	115
X1315	.032-.062	160	150	130	100	85
S.A.E. 1010-1025	.004-.015	160	140	110	90	80
3115-3130	.016-.031	140	110	90	80	60
4815-4820	.032-.062	110	90	80	70	45
S.A.E. 1030-1050	.004-.015	150	140	130	110	80
3130-3145	.016-.031	140	120	115	95	60
4825-5140	.032-.062	130	110	100	75	50
S.A.E. 1045	.004-.015	140	120	100	85	65
3240	.016-.031	120	100	80	75	50
4340-4640	.032-.062	100	85	65	60	45
S.A.E. 1050	.004-.015	120	100	80	75	60
S.A.E. 3135	.016-.031	100	80	70	65	50
S.A.E. 6125	.032-.062	80	70	60	50	40
S.A.E. 2340						
S.A.E. 1060-1085	.004-.015	120	100	80	75	50
3250	.016-.031	100	85	65	55	40
3340	.032-.062	80	75	50	45	30
9250						
S.A.E. 1095	.004-.015	100	90	75	65	50
3450	.016-.031	85	80	65	55	40
6130	.032-.062	75	70	55	45	30
Stainless	.004-.015	215	175	160	150	100
Steel (free	.016-.031	180	160	150	125	80
cutting)	.032-.062	160	140	130	100	60

NOTE. The cutting speeds given in the table are designed for short cuts and the use of coolants during the cutting action.

For continuous cuts without the use of a coolant, the cutting speeds should be reduced 15 to 20 per cent.

For intermittent cuts where a coolant is used, the cutting speeds should be decreased 15 to 20 per cent. Where no coolant is used, the speed should be reduced 20 to 25 per cent.

For light finishing cuts, where feeds are from .002" to .004" per revolution, the speed may be increased 40 to 80 per cent.

rake and clearance angles are shown in Fig. 11. Table VI gives the recommended angles for various materials. It should be remembered that these figures are merely starting points from which to work in either direction, depending upon the condition of the machine, the material, and all the other factors given heretofore.

Speeds and Feeds. Ordinarily, the feed for milling cutters in the roughing of cast iron, cast steel, or semisteel ranges from .006'' to .015'' chip load per tooth. Mild steel, however, can often be cut at .002'' to .010'' chip load, depending on the hardness of the work material and the power available.

On nonferrous metals and the plastics, it is often possible to increase the chip load to .030'' or more per tooth. How this chip load should be divided between speed, feed, and depth of cut is pretty much up to the tool engineer facing the specific problem. Much valuable information on these matters has been presented in foregoing chapters. Figures are given in Table VII which will go a long way in setting up the starting ranges of surface speed for various materials. As in the other tables in this book, the information presented merely represents points from which experimentation can be started for any specific job. The many factors that enter into on-the-job conditions are too variable to allow the setting up of hard and fast rules.

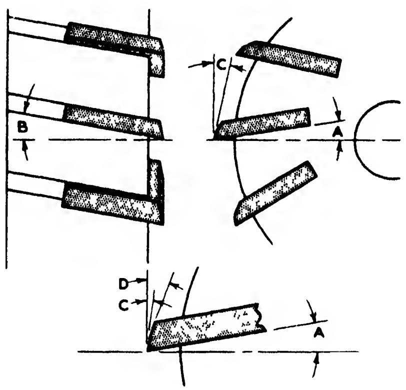


Fig. 11. Detail of Rakes and Clearances of Milling Cutter. *A* Is the Radial Rake, *B* the Axial, C Is the Primary Clearance, and the Secondary Clearance Is *D*

One of the great advantages of the cast alloys is their versatility. When there is a great variety of work, as in the jobbing shop, maintenance departments, and in smaller production shops where speed of machines are low, it may often be found economical to use cast alloy tools. Bronze, cast iron, aluminum, steel, or plastics can all be cut with the same tool, and with considerably more overall efficiency than normally would be possible with the sintered carbide where the grade of the carbide used is particularly important in any metal removal operation.

It is often found that the older type machines do not have enough speed or power to use the sintered carbides effectively. They may chatter and the bearings and gears may be badly worn. With the cast alloys, many of these machines can be used to capacity, particularly on small work. Many shops have found that combinations of the cast and sintered carbides do the best over-all job, the cast alloys being used for the smaller diameter work and the sintered carbides for the larger

TABLE VI. RECOMMENDED RAKES AND CLEARANCES FOR CAST ALLOY CUTTERS

Material	Axial Rake	Radial Rake	Primary Clearance	Secondary Clearance
Mild Steel....................	10	10	6	10
Cast Iron and Cast Steel..	0	6	6	10
Malleable Iron..............	0	6	6	10
Bronze........................	0	6	6	10
Aluminum....................	20	20	10	10

diameters at the same r.p.m., thus getting high speeds with large diameters and low speeds with the small.

In turret lathe work, for example, several tools may be engaged in one or successive operations at a fixed speed of the work. The tools will be cutting at high speed on the large diameters and at low speed on the smaller diameters. How this works is shown graphically in Fig. 12. Table VIII will prove helpful in making the proper selection of a tool in a case of this kind.

If the workpiece in Fig. 12 were a drop forging, for instance, and if sintered carbide tools were selected for the operation on the 10'' diameter, the speed of the 3'' diameter would be too low for effective use of the sintered carbide tools. If the 10'' cut were being made at 250 f.p.m., the 3'' diameter cut would be taken at only 75 f.p.m. It is then obvious that a combination of sintered carbide and cast alloy tools would be a simple and effective way to achieve satisfactory setup without stopping the machine for speed changes on the two cuts.

In machining the newer, heat-treated alloys, which are becoming increasingly common, the cast alloys often can turn in a performance that

TABLE VII. SURFACE SPEEDS FOR CAST ALLOY CUTTERS

Material Cut	Surface Speed in F.P.M.	
	Rough Milling	Finish Milling
Cast or Malleable Iron		
(roughing).....	80-140	150-250
(finishing).....	130-160	150-300
Cast Steel	60-80	80-100
Copper......................	400	600
Brass.........................	400	600
Bronze.......................	175	250
Aluminum	800	1,200 and up

the sintered carbides cannot equal. Sharp top and side rake angles are best for this type of work, but these design features cause chipping and

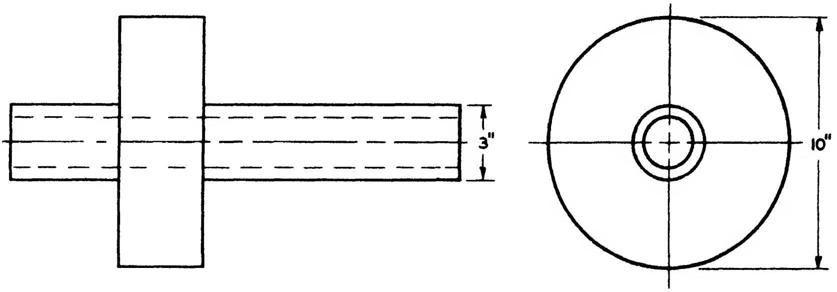


Fig. 12. An Example of a Piece of Work Which Probably Could Be Done Most Efficiently by Using a Combination of Carbide-Tipped and Cast Alloys Tools

excessive wear on the sintered carbide-tipped tools. The greater toughness of the cast alloys, especially those that include tantalum in their makeup, helps to support the cutting edges at these sharp angles. It is also believed that the inclusion of the tantalum helps reduce the total force in cutting because it allows the chips to slide better without pulling out particles of carbide from the tip.

TABLE VIII. COMPARATIVE CUTTING SPEEDS FOR TOOLS MADE OF DIFFERENT MATERIALS

Material Cut	Cutting Speed in f.p.m.				
	Carbon Steel	High-Speed Steel	Super-High-Speed Steel	Cast Alloys	Sintered Carbides
Aluminum.....	300-500	500-1000	800-1500	800-1500	1000-20,000
Brass...........	50-100	75-200	200-250	250-300	350-700
Bronze.........	40-75	75-150	150-200	175-250	250-500
Cast Iron					
Soft............	30-50	50-80	60-125	100-175	250-350
Hard..........	15-25	30-50	40-80	60-100	150-300
Chilled.......	....	15-20	30-50	40-75	100-250
Malleable....	30-50	70-100	80-125	125-200	250-400
Steel					
Soft............	30-50	60-100	75-150	175-250	250-400
Medium......	30-40	50-80	75-100	75-150	125-300
Hard..........	15-25	25-50	40-75	75-100	100-200

QUESTIONS TO HELP YOU

The following questions have been compiled as an aid in your reading. The important points in the chapter have been phrased interrogatively. If you are unable to answer any of them, it is suggested that you review the material just covered.

1. What is the range of wheel speeds used in grinding cast alloys?
2. Why should the speed not be lower than specified?
3. Why should the speed be no higher than specified?
4. With a narrow pointed tool, should the grinding feed be greater or less?
5. Are the wheels for grinding the cast alloys harder or softer than those used for high-speed steel?
6. Would the top face of a tool be ground if it is to be used on aluminum or brass?
7. Would the top face of the tool be ground if it is to be used on steel?
8. How many common types of chip breakers are there for single-pointed, cast alloy tools? Describe each one.
9. Why should a grinding wheel be diamond dressed before sharpening a milling cutter blade?
10. How much should the secondary clearance be and how is it ground?
11. What is the "quadrant" method of grinding milling cutters? What are its advantages?
12. If it was necessary to finish turn a steel casting having two diameters, one of 9", the other of 2 1/2", and chamfer one end (2 1/2" dia.) 45°, all in one tooling and at a work speed of 125 r.p.m., what materials should be specified for the tools?

REFERENCES

Haynes Stellite Co. Haynes Stellite Metal Cutting Tools. Kokomo, Indiana: Haynes Stellite Co., February 1, 1945.

Smith, E. F. "Chip Breakers for Stellite Tools," American Machinist, January 11, 1939.

———. "Saving Time with Smaller Tools," American Machinist, February 8, 1939.

Vascoloy-Ramet Corp. Tantung Tool Catalog. North Chicago, Ill.: Vascoloy-Ramet Corp., 1943.

BIBLIOGRAPHY

Bily, C. "Adjustable-Wedge Toolholder for High-Load Carbide Turning," American Machinist, 90:138 (August 15, 1946).

Boston, O. W., and Gilbert, W. W. "Effect of Varying Relief Angles When Face Milling Cast Iron with Sintered Carbide-Tipped Cutters," A.S.M.E. Trans., 69:657-63 (August, 1947).

Dinger, F. R. "Roll Turning with Carbide Tools," Iron and Steel Engineering, 25:66-9 (September, 1948).

Dym, J. B., and Badger, T. "Cast Alloys Vary in Cutting Efficiency," American Machinist, 91:109-11 (November 20, 1947).

Eckersley, H. "Manufacture and Application of Sintered Carbides," Engineering, 184:420-2 (October 31, 1947).

Field, M. "Practice and Theory in Carbide Milling," Steel, 121:100 (September 29, 1947).

Frommelt, H. A. "Reconditioning Cutting Tools for Maximum Efficiency," Steel, 121:102-5 (November 10, 1947).

Gumpper, J. C., and Badger, T. "How Westinghouse Standardized Single Point Carbide Tools," Machinery, 54:162-7 (March, 1948).

Halloway, C. W. "Cushioned Tips Extend Life of Negative-Rake Turning Tools," American Machinist, 90:108-9 (January 31, 1946).

Hirschfeld, E. "Carbide Requires More Powerful Machines," American Machinist, 90:108-2 (September 26, 1946).

Kronenberg, M. "Analysis of Initial Contact of Milling Cutter and Work in Relation to Tool Life," A.S.M.E. Trans., 68:217-28 (April, 1946).

Maurer, R. G. "Removing Carbide Tips from Tool Shanks," Metal Progress, 49:537 (March, 1946).

Raible, G. J. "Planing Cast Iron with Carbide Tools," Machinery, 55:171-3 (September, 1948).

Safhill, J. "Fitting Carbides to Small Tools," American Machinist, 91:117 (October 9, 1947).

Sharp, G. W. "Better Method for Sharpening Carbide Face Mills," American Machinist, 91:108 (March 13, 1947).

Wick, C. H. "Sharpening Carbide Tools," Machinery, 54:145-51 (June, 1948).

York, H. E. "All-Carbide Boring Bar," American Machinist, 90:115 (May 23, 1946).

INDEX

Useful Information

To find the circumference of a circle, multiply the diameter by 3.1416.

To find the diameter of a circle, multiply the circumference by .31831.

To find the area of a circle, multiply the square of the diameter by .7854.

The radius of a circle × 6.283185 = the circumference.

The square of the circumference of a circle × .07958 = the area.

Half the circumference of a circle × half its diameter = the area.

The circumference of a circle × .159155 = the radius.

The square root of the area of a circle × .56419 = the radius.

The square root of the area of a circle × 1.12838 = the diameter.

To find the diameter of a circle equal in area to a given square, multiply a side of the square by 1.12838.

To find the side of a square equal in area to a given circle, multiply the diameter by .8862.

To find the side of a square inscribed in a circle, multiply the diameter by .7071.

To find the side of a hexagon inscribed in a circle, multiply the diameter of the circle by .500.

To find the diameter of a circle inscribed in a hexagon, multiply a side of the hexagon by 1.7321.

To find the side of an equilateral triangle inscribed in a circle, multiply the diameter of the circle by .866.

To find the diameter of a circle inscribed in an equilateral triangle, multiply a side of the triangle by .57735.

To find the area of the surface of a ball (sphere), multiply the square of the diameter by 3.1416.

To find the volume of a ball (sphere), multiply the cube of the diameter by .5236.

Doubling the diameter of a pipe increases its capacity four times.

To find the pressure in pounds per square inch at the base of a column of water, multiply the height of the column in feet by .433.

A gallon of water (U. S. Standard) weighs 8.336 pounds and contains 231 cubic inches. A cubic foot of water contains 7½ gallons, 1728 cubic inches, and weighs 62.425 pounds at a temperature of about 39° F.

These weights change slightly above and below this temperature.

In accordance with the standard practice approved by the American Standards Association, the ratio 25.4 mm = 1 inch is used for converting millimeters to inches. This factor varies only two millionths of an inch from the more exact factor 25.40005 mm, a difference so small as to be negligible for industrial length measurements.

Metric Measures

The metric unit of length is the meter = 39.37 inches.

The metric unit of weight is the gram = 15.432 grains.

The following prefixes are used for sub-divisions and multiples: Milli = $\frac{1}{1000}$, Centi = $\frac{1}{100}$, Deci = $\frac{1}{10}$, Deca = 10, Hecto = 100, Kilo = 1000, Myria = 10,000.

Metric and English Equivalent Measures

MEASURES OF LENGTH

Metric		*English*
1 meter	=	39.37 inches, or 3.28083 feet, or 1.09361 yards
.3048 meter	=	1 foot
1 centimeter	=	.3937 inch
2.54 centimeters	=	1 inch
1 millimeter	=	.03937 inch, or nearly 1-25 inch
25.4 millimeters	=	1 inch
1 kilometer	=	1093.61 yards, or 0.62137 mile

MEASURES OF WEIGHT

Metric		*English*
1 gram	=	15.432 grains
.0648 gram	=	1 grain
28.35 grams	=	1 ounce avoirdupois
1 kilogram	=	2.2046 pounds
.4536 kilogram	=	1 pound
1 metric ton 1000 kilograms	=	.9842 ton of 2240 pounds 19.68 cwt. 2204.6 pounds
1.016 metric tons 1016 kilograms	=	1 ton of 2240 pounds

MEASURES OF CAPACITY

Metric		*English*
1 liter (= 1 cubic decimeter)	=	61.023 cubic inches .03531 cubic foot .2642 gal. (American) 2.202 lbs. of water at 62° F.
28.317 liters	=	1 cubic foot
3.785 liters	=	1 gallon (American)
4.543 liters	=	1 gallon (Imperial)

English Conversion Table

Length			
Inches	×	.0833	= feet
Inches	×	.02778	= yards
Inches	×	.00001578	= miles
Feet	×	.3333	= yards
Feet	×	.0001894	= miles
Yards	×	36.00	= inches
Yards	×	3.00	= feet
Yards	×	.0005681	= miles
Miles	×	63360.00	= inches
Miles	×	5280.00	= feet
Miles	×	1760.00	= yards
Circumference of circle	×	.3188	= diameter
Diameter of circle	×	3.1416	= circumference
Area			
Square inches	×	.00694	= square feet
Square inches	×	.0007716	= square yards
Square feet	×	144.00	= square inches
Square feet	×	.11111	= square yards
Square yards	×	1296.00	= square inches
Square yards	×	9.00	= square feet
Dia. of circle squared	×	.7854	= area
Dia. of sphere squared	×	3.1416	= surface
Volume			
Cubic inches	×	.0005787	= cubic feet
Cubic inches	×	.00002143	= cubic yards
Cubic inches	×	.004329	= U. S. gallons
Cubic feet	×	1728.00	= cubic inches
Cubic feet	×	.03704	= cubic yards
Cubic feet	×	7.4805	= U. S. gallons
Cubic yards	×	46656.00	= cubic inches
Cubic yards	×	27.00	= cubic feet
Dia. of sphere cubed	×	.5236	= volume
Weight			
Grains (avoirdupois)	×	.002286	= ounces
Ounces (avoirdupois)	×	.0625	= pounds
Ounces (avoirdupois)	×	.00003125	= tons
Pounds (avoirdupois)	×	16.00	= ounces
Pounds (avoirdupois)	×	.01	= hundredweight
Pounds (avoirdupois)	×	.0005	= tons
Tons (avoirdupois)	×	32000.00	= ounces
Tons (avoirdupois)	×	2000.00	= pounds

English Conversion Table

Energy			
Horsepower	×	33000.	= ft.-lbs. per min.
B. t. u.	×	778.26	= ft.-lbs.
Ton of refrigeration	×	200.	= B. t. u. per min.
Pressure			
Lbs. per sq. in.	×	2.31	= ft. of water (60°F.)
Ft .of water (60°F.)	×	.433	= lbs. per sq. in.
Ins. of water (60°F.)	×	.0361	= lbs. per sq. in.
Lbs. per sq. in.	×	27.70	= ins. of water (60°F.)
Lbs. per sq. in.	×	2.041	= ins. of Hg. (60°F.)
Ins. of Hg (60°F.)	×	.490	= lbs. per sq. in.
Power			
Horsepower	×	746.	= watts
Watts	×	.001341	= horsepower
Horsepower	×	42.4	= B. t. u. per min.
***Water Factors* (at point of greatest density—39.2°F)**			
Miners inch (of water)	×	8.976	= U. S .gals. per min.
Cubic inches (of water)	×	.57798	= ounces
Cubic inches (of water)	×	.036124	= pounds
Cubic inches (of water)	×	.004329	= U. S. gallons
Cubic inches (of water)	×	.003607	= English gallons
Cubic feet (of water)	×	62.425	= pounds
Cubic feet (of water)	×	.03121	= tons
Cubic feet (of water)	×	7.4805	= U. S. gallons
Cubic feet (of water)	×	6.232	= English gallons
Cubic foot of ice	×	57.2	= pounds
Ounces (of water)	×	1.73	= cubic inches
Pounds (of water)	×	26.68	= cubic inches
Pounds (of water)	×	.01602	= cubic feet
Pounds (of water)	×	.1198	= U. S. gallons
Pounds (of water)	×	.0998	= English gallons
Tons (of water)	×	32.04	= cubic feet
Tons (of water)	×	239.6	= U. S. gallons
Tons (of water)	×	199.6	= English gallons
U. S. gallons	×	231.00	= cubic inches
U. S. gallons	×	.13368	= cubic feet
U. S. gallons	×	8.345	= pounds
U. S. gallons	×	.8327	= English gallons
U. S. gallons	×	3.785	= liters
English gallons (Imperial)	×	277.41	= cubic inches
English gallons (Imperial)	×	.1605	= cubic feet
English gallons (Imperial)	×	10.02	= pounds
English gallons (Imperial)	×	1.201	= U. S. gallons
English gallons (Imperial)	×	4.546	= liters

Metric Conversion Table

Length

Millimeters	×	.03937	= inches
Millimeters	÷	25.4	= inches
Centimeters	×	.3937	= inches
Centimeters	÷	2.54	= inches
Meters	×	39.37	= inches (Act. Cong.)
Meters	×	3.281	= feet
Meters	×	1.0936	= yards
Kilometers	×	.6214	= miles
Kilometers	÷	1.6093	= miles
Kilometers	×	3280.8	= feet

Area

Sq. Millimeters	×	.00155	= sq. in.
Sq. Millimeters	÷	645.2	= sq. in.
Sq. Centimeters	×	.155	= sq. in.
Sq. Centimeters	÷	6.452	= sq. in.
Sq. Meters	×	10.764	= sq. ft.
Sq. Kilometers	×	247.1	= acres
Hectares	×	2.471	= acres

Volume

Cu. Centimeters	÷	16.387	= cu. in.
Cu. Centimeters	÷	3.69	= fl. drs. (U.S.P.)
Cu. Centimeters	÷	29.57	= fl. oz. (U.S.P.)
Cu. Meters	×	35.314	= cu. ft.
Cu. Meters	×	1.308	= cu. yards
Cu. Meters	×	264.2	= gals. (231 cu. in.)
Litres	×	61.023	= cu. in. (Act. Cong.)
Litres	×	33.82	= fl. oz. (U.S.P.)
Litres	×	.2642	= gals. (231 cu. in.)
Litres	÷	3.785	= gals. (231 cu. in.)
Litres	÷	28.317	= cu. ft.
Hectolitres	×	3.531	= cu. ft.
Hectolitres	×	2.838	= bu. (2150.42 cu. in.)
Hectolitres	×	.1308	= cu. yds.
Hectolitres	×	26.42	= gals. (231 cu. in.)

Weight

Grams	×	15.432	= grains (Act. Cong.)
Grams	÷	981.	= dynes
Grams (water)	÷	29.57	= fl. oz.
Grams	÷	28.35	= oz. avoirdupois
Kilo-grams	×	2.2046	= lbs.

Metric Conversion Table (Cont.)

Weight

Kilo-grams	×	35 27	= oz. avoirdupois
Kilo-grams	×	0011023	= tons (2000 lbs.)
Tonneau (Metric ton)	×	1 1023	= tons (2000 lbs.)
Tonneau (Metric ton)	×	2204 6	= lbs.

Unit Weight

Grams per cu. cent.	÷	27 68	= lbs per cu. in.
Kilo per meter	×	672	= lbs per ft.
Kilo per cu meter	×	06243	= lbs per cu. ft.
Kilo per Cheval	×	2 235	= lbs per h p.
Grams per liter	×	.06243	= lbs per cu ft.

Pressure

Kilo-grams per sq. cm.	×	14 223	= lbs per sq in.
Kilo-grams per sq cm.	×	32 843	= ft of water (60°F.)
Atmospheres (international)	×	14 696	= lbs per sq in

Energy

Joule	×	7376	= ft lbs.
Kilo-gram meters	×	7 233	= ft lbs.

Power

Cheval vapeur	×	9863	= h p.
Kilo-watts	×	1 341	= h p.
Watts	÷	746	= h p.
Watts	×	7373	= ft lbs per sec

Miscellaneous

Kilogram calorie	×	3 968	= B. t. u.
Standard gravity (Sea level 45° lat.)	÷	980.665	= centimeters per sec. per sec.
Frigories/hr. (French)	÷	3023.9	= Tons refrigeration

The following pages show temperatures on Fahrenheit and Centigrade thermometers.

Equivalent Temperature Readings for Fahrenheit and Centigrade Scales

Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.
—459.4	—273	—21.	—29.4	17.6	—8.	56.	13.3
—436.	—270.	—20.2	—29.	18.	—7.8	57.	13.9
—418.	—260.	—20.	—28.9	19.	—7.2	57.2	14.
—400.	—240.	—19.	—28.3	19.4	—7.	58.	14.4
—382.	—230.	—18.4	—28.	20.	—6.7	59.	15.
—364.	—220.	—18.	—27.8	21.	—6.1	60.	15.6
—346.	—210.	—17.	—27.2	21.2	—6.	60.8	16.
—328.	—200.	—16.6	—27.	22.	—5.6	61.	16.1
—310.	—190.	—16.	—26.7	23.	—5.	62.	16.7
—292.	—180.	—15.	—26.1	24.	—4.4	62.6	17.
—274.	—170.	—14.8	—26.	24.8	—4.	63.	17.2
—256.	—160.	—14.	—25.6	25.	—3.9	64.	17.8
—238.	—150.	—13.	—25.	26.	—3.3	64.4	18.
—220.	—140	—12.	—24.4	26.6	—3.	65.	18.3
—202.	—130.	—11.2	—24.	27.	—2.8	66.	18.9
—184.	—120.	—11.	—23.9	28.	—2.2	66.2	19.
—166.	—110.	—10.	—23.3	28.4	—2.	67.	19.4
—148.	—100.	— 9.4	—23.	29.	—1.7	68.	20.
—139.	— 95.	— 9.	—22.8	30.	—1.1	69.	20.6
—130.	— 90.	— 8.	—22.2	30.2	—1.	69.8	21.
—121.	— 85.	— 7.6	—22.	31.	—0.6	70.	21.1
—112	— 80.	— 7.	—21.7	32.	0.	71.	21.7
—103.	— 75	— 6.	—21.1	33.	+0.6	71.6	22.
— 94.	— 70.	— 5.8	—21.	33.8	1.	72.	22.2
— 85.	— 65.	— 5.	—20.6	34.	1.1	73.	22.8
— 76.	— 60.	— 4.	—20.	35.	1.7	73.4	23.
— 67.	— 55.	— 3.	—19.4	35.6	2.	74.	23.3
— 58.	— 50.	— 2.2	—19.	36.	2.2	75.	23.9
— 49.	— 45.	— 2.	—18.9	37.	2.8	75.2	24.
— 40.	— 40.	— 1.	—18.3	37.4	3.	76.	24.4
— 39.	— 39.4	— 0.4	—18.	38.	3.3	77.	25.
— 38.2	— 39.	0.	—17.8	39.	3.9	78.	25.6
— 38.	— 38.9	+ 1.	—17.2	39.2	4.	78.8	26.
— 37.	— 38.3	1.4	—17.	40.	4.4	79.	26.1
— 36.4	— 38.	2.	—16.7	41.	5.	80.	26.7
— 36.	— 37.8	3.	—16.1	42.	5.6	80.6	27.
— 35.	— 37.2	3.2	—16.	42.8	6.	81.	27.2
— 34.6	— 37.	4.	—15.6	43.	6.1	82.	27.8
— 34.	— 36.7	5.	—15.	44.	6.7	82.4	28.
— 33.	— 36.1	6.	—14.4	44.6	7.	83.	28.3
— 32.8	— 36.	6.8	—14.	45.	7.2	84.	28.9
— 32.	— 35.6	7.	—13.9	46.	7.8	84.2	29.
— 31.	— 35.	8.	—13.3	46.4	8.	85.	29.4
— 30.	— 34.4	8.6	—13.	47.	8.3	86.	30.
— 29.2	— 34.	9.	—12.8	48.	8.9	87.	30.6
— 29.	— 33.9	10.	—12.2	48.2	9.	87.8	31.
— 28.	— 33.3	10.4	—12.	49.	9.4	88.	31.1
— 27.4	— 33.	11.	—11.7	50.	10.	89.	31.7
— 27.	— 32.8	12.	—11.1	51.	10.6	89.6	32.
— 26.	— 32 2	12.2	—11.	51.8	11.	90.	32.2
— 25.6	— 32.	13.	—10.6	52.	11.1	91.	32.8
— 25.	— 31.7	14.	—10.	53.	11.7	91.4	33.
— 24.	— 31.1	15.	— 9.4	53.6	12.	92.	33.3
— 23.8	— 31.	15.8	— 9.	54.	12.2	93.	33.9
— 23.	— 30.6	16.	— 8.9	55.	12.8	93.2	34.
— 22.	— 30.	17.	— 8.3	55.4	13.	94.	34.4

Equivalent Temperature Readings for Fahrenheit and Centigrade Scales

Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.	Fahrenheit Degs.	Centigrade Degs.
95.	35.	134.	56 7	172 4	78.	211.	99.4
96.	35 6	134 6	57.	173.	78 3	212.	100.
96 8	36.	135.	57 2	174.	78 9	213.	100 6
97.	36 1	136.	57 8	174 2	79.	213 8	101.
98.	36.7	136 4	58.	175.	79 4	214.	101 1
98.6	37.	137.	58 3	176.	80.	215.	101.7
99.	37 2	138.	58 9	177.	80 6	215 6	102.
100.	37 8	138 2	59.	177 8	81.	216.	102 2
100 4	38.	139.	59 4	178.	81 1	217.	102.8
101.	38 3	140.	60.	179.	81 7	217.4	103.
102.	38 9	141.	60 6	179 6	82.	218.	103 3
102 2	39.	141 8	61.	180.	82 2	219.	103 9
103.	39 4	142.	61 1	181.	82 8	219 2	104.
104.	40.	143.	61 7	181 4	83.	220.	104 4
105.	40 6	143 6	62.	182.	83 3	221.	105.
105 8	41.	144.	62 2	183.	83 9	222.	105 6
106.	41 1	145.	62 8	183 2	84.	222 8	106.
107.	41 7	145 4	63.	184.	84 4	223.	106 1
107 6	42.	146.	63 3	185.	85.	224.	106 7
108.	42 2	147.	63 9	186.	85 6	224 6	107.
109.	42 8	147 2	64.	186 8	86.	225.	107 2
109 4	43.	148.	64 4	187.	86 1	226.	107 8
110.	43 3	149.	65.	188.	86 7	226 4	108.
111.	43 9	150.	65 6	188 6	87.	227.	108 3
111 2	44.	150 8	66.	189.	87 2	228.	108 9
112.	44 4	151.	66 1	190.	87 8	228 2	109.
113.	45.	152.	66 7	190 4	88.	229.	109 4
114.	45 6	152 6	67.	191.	88 3	230.	110.
114 8	46.	153.	67 2	192.	88 9	231.	110 6
115.	46 1	154.	67 8	192 2	89.	231 8	111.
116.	46 7	154 4	68.	193.	89 4	232.	111 1
116 6	47.	155.	68 3	194.	90.	233.	111 7
117.	47 2	156.	68 9	195.	90 6	233 6	112.
118.	47 8	156 2	69.	195 8	91.	234.	112 3
118 4	48.	157.	69 4	196.	91 1	235.	112 8
119.	48.3	158.	70.	197.	91 7	235 4	113.
120.	48 9	159.	70 6	197 6	92.	236.	113 3
120 2	49.	159 8	71.	198.	92 2	237.	113.9
121.	49 4	160.	71 1	199.	92 8	237 2	114.
122.	50.	161.	71 7	199 4	93.	238.	114 4
123.	50 6	161 6	72.	200.	93 3	239.	115.
123 8	51.	162.	72 2	201.	93 9	240.	115 6
124.	51 1	163.	72 8	201 2	94.	240 8	116.
125.	51 7	163 4	73.	202.	94 4	241.	116 1
125 6	52.	164.	73 3	203.	95.	242.	116 7
126.	52 2	165.	73 9	204.	95 6	242 6	117.
127.	52 8	165 2	74.	204 8	96.	243.	117 2
127 4	53.	166.	74 4	205.	96 1	244.	117 8
128.	53 3	167.	75.	206.	96 7	244 4	118.
129.	53 9	168.	75 6	206 6	97.	245.	118 3
129.2	54.	168 8	76.	207.	97 2	246.	118 9
130.	54 4	169.	76 1	208.	97.8	246 2	119.
131.	55.	170.	76 7	208 4	98.	247.	119.4
132.	55 6	170 6	77.	209.	98 3	248.	120.
132 8	56.	171.	77.2	210.	98.9	249.	120 6
133.	56.1	172.	77.8	210 2	99.	249.8	121.